VENABLE WINDOWS VERSION 6.3 SOFTWARE AND HARDWARE MANUAL

FOR

SERIES 43xx/51xx/63xx/74xx/88xx/350c SYSTEMS

Venable Instruments

**8656 SH 71 West, Bldg. E
Cuesta Centre
Austin, TX 78735**

512-949-3100

http://www.venableinstruments.com/

TABLE OF CONTENTS

TOPIC   **PAGE**

Introduction

This manual describes the Venable Windows Software that is part of Venable's complete frequency response modeling and measurement system.

This manual is provided in printed form with the hardware Analyzer, in electronic form in the software and in electronic form on the download site. The latest version of this manual can always be found on Venable's download site beside the current software installation package. The version in the current software installation can be reached using the menu item Help → Venable Manual.

The installation portion of this manual covers both the hardware and software setup.

The Software works with a number of different hardware models of Frequency Response Analyzers. Separate licenses must be obtained from Venable for each type of Analyzer the Software will work with. Without a license, the software will read saved files and allow you to view the data and graphs but not gather new data or save files.

The Software adapts to the differences between Analyzers by showing different Analyzer Control dialogs and making minor changes in other dialogs. The main Software Window will display the currently connected _analyzer_ as part of the window title in the title bar.

This manual will point out the differences between Analyzer types where it matters. Type specific screenshots of several dialogs are given; please look at the information relevant to the type of Analyzer you are working with.

System Overview

The Venable Windows Software in combination with a supported Frequency Response Analyzer (FRA) is a complete frequency response modeling and measurement system.

The hardware portion consists of a FRA, which is used for making measurements of gain, phase, and voltage versus frequency and various accessories for coupling the FRA to the electrical, mechanical, or thermal system under test. Supported FRA's are the following[1]. The list is grouped by Software license type:

- 4000 for a Venable Series 43xx/63xx/74xx/88xx/350c System
- 4000RLC for a Venable Series 43xx/63xx/74xx/88xx/350c System
- 5140 for a Venable Series 5140 System
- 5140RLC for a Venable Series 5140 System

The FRA is controlled by this software using a standard USB connection.

The software portion runs on any personal computer using Microsoft Windows 7, 8 or 10, 32-bit or 64-bit.

The Error Amplifier Synthesis and Circuit Analysis menu and the Math menu can be enabled for the 5140. These two menus are standard for all our other models.

The analysis menu contains a simple spice-like modeling program for modeling the AC frequency response of circuits. Model results and test results are in the same format and can be displayed simultaneously for easy comparison. The error amplifier synthesis software lets the user achieve the exact feedback loop bandwidth and phase margin desired on the first try.

The Math menu allows any kind of mathematical function on any one or two transfer functions.

The Venable RLC software has been incorporated as menu in the new Venable version 6.1 software. It can be enabled through a separate RLC license. You can calibrate your RLC fixture, run impedance measurements through a range of frequencies, build a theoretical model based on the empirical data, and generate reports.

Data can be saved, recalled, and printed in the form of graphs on any Windows-compatible printer in black-and-white or color if the printer has that capability.

The Venable software is protected against illegal copying and use. Anyone can install the software to read and view data files. In order to modify data files or connect to an analyzer, a matching license for the analyzer type must be activated. This is done through the menu item: Analyzers -> Licenses.

04/14/20

Installation Procedure

This software uses the USB port on the back of the analyzer to communication with the PC. This is the step-by-step software and hardware installation procedure.

1. Close all your programs and insert the Venable installation disk into your CD-ROM drive. The CD should auto-run and the installation window should open up, otherwise browse the CD with Windows Explorer and double-click on VenableSetup.exe. Follow the prompts to allow the installer to install the Venable Software to your PC. The correct driver will be setup under Windows 8 and 10. For some Windows 7 installations, you will need to follow the prompts to install WinUSB from Windows Update. See below for driver installation instructions.

    2. Make sure you have your software serial number at hand and follow the instructions.

        The Venable software is protected against illegal copying and use. In order to connect to analyzers and modify data files, each type of analyzer must be activated by the Licensing Wizard built in to the Venable software. From the software main menu, the wizard is accessible from Analyzers -> Licenses.

        Manuals for the Venable software, Manual.pdf, and the Licensing Software, LicenseManual.pdf, are included in the installation directory. These manuals are in Adobe portable document format (pdf) and are compatible with Adobe Acrobat Reader. From the Venable software program you can open the manuals using the menu items "Help -> Venable Manual" and "Help -> License Manual".

        If you need to uninstall the software for any reason, use Add/Remove Programs and choose the uninstall option.

Verify the Installation

1. Connect a standard USB cable to the analyzer. Plug the other end of the USB cable into a USB port on the PC. Turn on the Venable FRA analyzer. Wait for analyzer to complete startup, approximately 15 seconds. Under Windows 7, you may be prompted to find the missing driver, allow Windows to find the best driver from Windows Update.

2. Go to the Windows System Tray and click on the USB icon. You should see Eject Frequency Response Analyzer if the Windows USB driver has loaded. If not, go to the following section on WinUSB Driver Installation Troubleshooting.
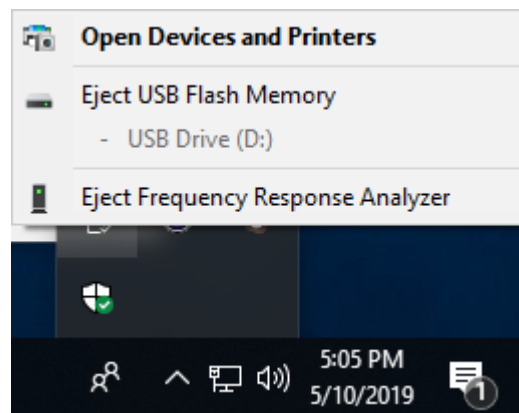
System Tray

3. If the driver has loaded, open the Venable software. The Analyzer Control, Readings and RLC Control (option) docking windows will be inactive. If these windows aren't visible go to the View menu and make them visible by selecting them.
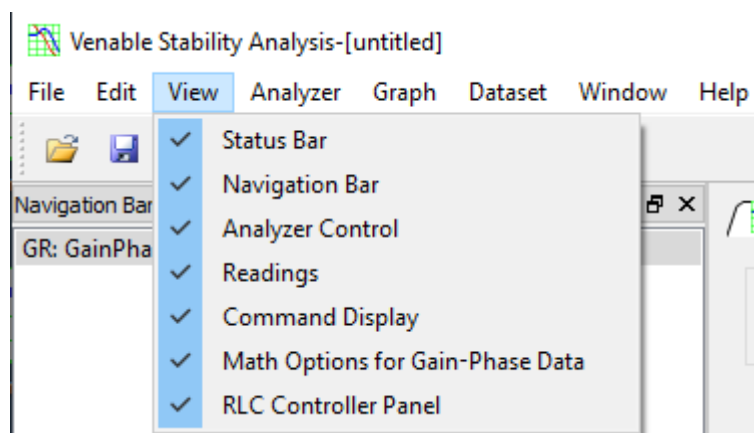
View Menu

4. Select the Analyzer menu in the Venable Software. This will display a list of available analyzers to connect to. Select the analyzer from the list. The Venable software will automatically connect to the selected analyzer and the Analyzer Control, Readings and RLC Control (option) docking windows will become active.
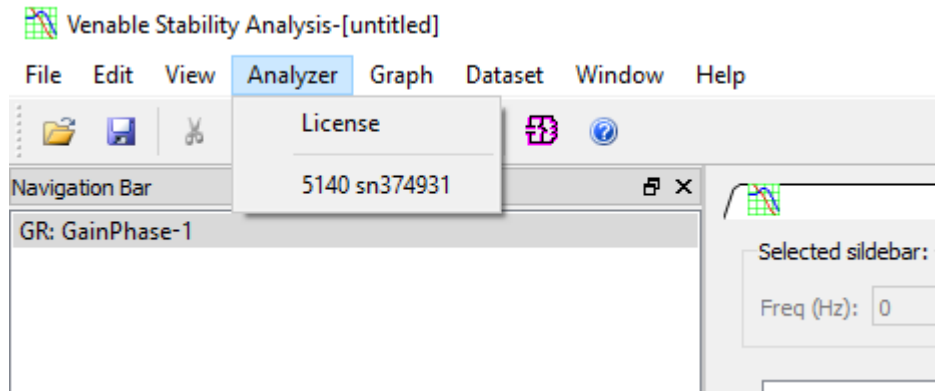
Analyzer Menu

If the analyzer in the Analyzer menu is dimmed out then you have not activated a license for that analyzer type yet. If the analyzer does not show up in the list then check the WinUSB Driver Installation Troubleshooting to ensure the driver is correct.

This is a common problem when upgrading the software from version 5 to version 6. The system might be loading the Agilent/Keysight USB driver for the version 5 software instead of the WinUSB driver for version 6 and/or the registry entry for the driver is incorrect.

WinUSB Driver Installation Troubleshooting

Analyzers running communication firmware version 3.0 or later (typically built after Jan. 2019) normally identify themselves to Windows as using the WinUSB driver instead of the Keysight IO drivers. This driver is built-in to Windows 8 and 10 and is used automatically. Under Windows 7, you might need to install the driver over the Internet from Windows Update if you are prompted on the first time you plug in a WinUSB capable analyzer.

1. You can check that the correct driver is being used by going to the Control Panel and opening the Device Manager. Look for a WinUSB device named "Venable Frequency Response Analyzer" under the Universal Serial Bus Devices.

2. In Windows 7, other drivers for USB Test and Measurement IVI devices can grab the analyzer by mistake. This can happen with the Agilent/Keysight IO Suite drivers used by previous Venable software.

Agilent/Keysight Driver

3. If you find the analyzer listed as a Test and Measurement IVI device under the USB Test and Measurement Devices category, then do the following to repair this issue:

   A. Open Device Manager from the Control Panel. Find "USB Test and Measurement Device (IVI). Double click on this device to open the properties dialog. Select the Driver tab and click on "Update Driver". Click on "Browse my computer for driver software".

   B. Click on "Let me pick from a list of available drivers on my computer". Select the "WinUsb Device" and click on "Next".
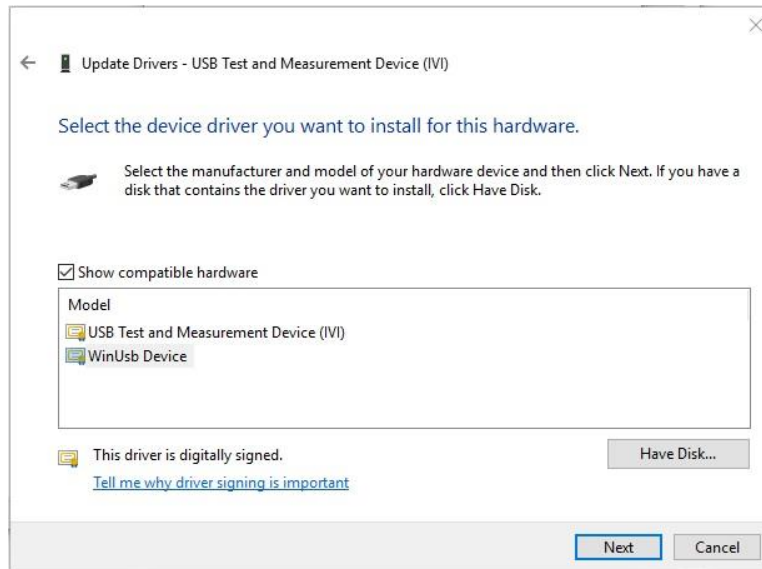
List of Available Device Drivers

C. Wait for message that Windows has successfully updated your drivers. Follow remaining instructions then cycle power on the analyzer and restart the Venable software.

4. If the driver is correct or manually changing the driver still doesn't fix the problem, registry entries may be preventing the driver from loading correctly. You will have to run a batch file called SetDriverToVer6.bat to fix the registry. This file is located in the tools folder of the Venable software installation disk or can be downloaded from the Venable Instruments website. You will need to run this program in Administrator mode in order to fix the issue.

5. You will need to cycle the power on the analyzer in order for the driver change to complete. Once the analyzer boot up completes, the new driver association can be verified in the Device Manager.

04/14/20

"Sanity check" – Measure the Output of the Oscillator

After you have connected the hardware and turned the power on and installed the software, a natural question that comes up is "Does it work?" To answer that question, just follow these simple steps:

1. Hook up the BNC to Mini-grabber cables that come with the system to the oscillator output and to each of the inputs of the FRA. Connect all the black (-) Mini-grabbers together and all the red (+) Mini-grabbers together.

2. The Analyzer Control panel should no longer be grayed out once the software has made the communication connection to the analyzer.

3. Set the Start Frequency to 10 Hz. Set the AC Volts Out to 1 Volt peak. Click the Take Data at Start Frequency. The Readings window should then display the following values based on analyzer type:

Analyzer type	Frequency (Hz)	2 channels (Vrms)	Gain (dB)	Phase (degrees)
43xx/51xx/63xx/74xx/88xx/350c	10	0.707	0	0

Data taken by the analyzer will be displayed in the Readings window and communication traffic will be displayed in the Command Display window.

Analyzer Control

Turn On DC voltage

| Stop | Run Sweep |

Start Freq (Hz): 10

Stop Freq (Hz): 1 M

Log Sweep (Steps/Decade): 10

AC Volts Out (VAC peak): 1

DC Volts Out (VDC): 0

Waveform: sine

Output Imp (Ohm): 50

Servo Control

☐ Servo On: CH1

AC Volts In (Vrms): 10 m

AC Out Step Size (dB): 10

Max AC Volts Out (Vpk): 10

Integration

Time (sec): 1

Delay: 0

Input Imp (Ohm): 1e6

Maximum Change Between Data Points

☐ Gain (dB): 10

☐ Phase (Deg): 20

Option

☐ 350b operating Mode   ☐ Sync Output

☐ High Noise Algorithm   ☐ Digital Interface

Circuit Under Test: DC

Analyzer Control Window

Readings

Freq. (Hz): 10   Ratio: CH2/CH1

CH1: 709.038 m   CH2: 709.239 m

Gain (dB): 2.4693 m   Phase (deg): -7.23559 m

Readings Window

If this all works, the system is hooked up and working properly.

Tutorial No.1 – Measure an RC Low-Pass Filter Transfer Function

If the "Sanity check" worked out but you still are not convinced, try this as a real-life application of a frequency response analyzer:

1. Take a resistor and capacitor you know the value of and solder them in series. Using the same BNC-Mini-grabber cables as before, connect both the oscillator output and the main input of channel 1 across both parts with the black (return) lead connected to the capacitor. Connect channel 2 across the capacitor with the black (return) lead connected the same place as the other two black leads. Calculate the corner frequency of the filter from the formula $f=1/2\pi RC$. Or just use a 1k resistor and a 0.1 µF capacitor and the corner frequency will be 1.6 kHz.

2. Set the sweep from approximately 2 decades below the corner frequency to approximately 2 decades above the corner frequency (10 Hz to 100 kHz if you use 1k and 0.1 µF).

   Select Log Sweep, 10 Steps per Decade, 1.414 volts peak AC, 0 volts DC, sine wave, 50 Ohm output impedance, 1-second integration, 0 delay time and 1 MOhm input impedance.

   Click on Run Sweep.

3. The Data Set Properties sub window gives you the choice of channel ratios to plot to the graph. You can also set the scale factor of the plot. The default settings CH2/CH1 and Scale Factor=1 are correct for this measurement.

4. The sweep will start. You can see the plot of the data being taken in real time. The resulting gain plot should be flat at 0 dB out to the corner frequency, then fall at a -20 dB per decade slope thereafter. The phase should be 0 degrees at low frequency, -45 degrees at the corner, and asymptotically approach -90 degrees at high frequency. If this works, you can be certain that the equipment is connected and functioning properly.
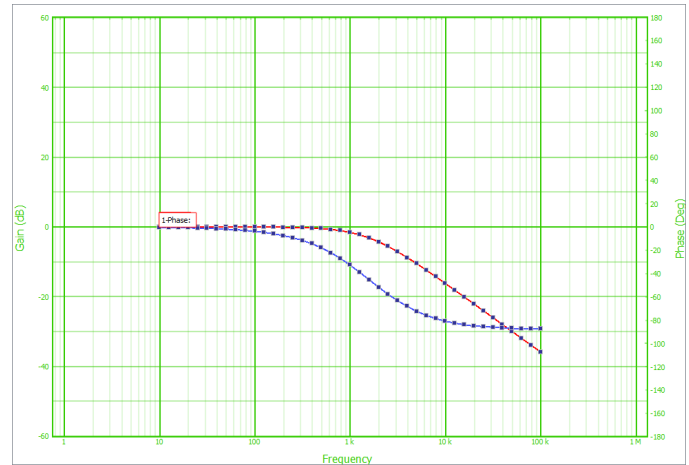
Measured transfer function of
low pass R-C filter

Tutorial No.2 – Model of an RC Low-Pass Filter Transfer Function

As a simple example of how the modeling portion of the software works, an easy thing to do is model the transfer function of the low-pass RC filter corresponding to the test in Tutorial No. 1. If you actually did the test, you can overlay the test and model data to see how close you came.

1. To start the process, click on the "Error Amps and Circuits" Icon, the one just to the left of the Analyzer Control Icon. When that window opens, select the "Circuit (CKT) Model" tab. If there is anything in the text box, delete it and type in the text below:

2.

```
V 0 0 1
R 1 0 1
R 1 2 1K
C 2 0 0.1U
```
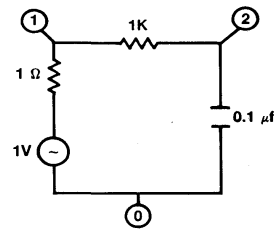
3. For the record, the first two lines represent a fixed voltage source with a magnitude of 1 volt and an internal resistance of 1 ohm, connected from node 1 (the input) to node 0 (ground). The third line represents a 1k resistor connected from node 1 (the input) to node 2 (the output). The final line, line 4, represents a capacitor connected from node 2 (the output) to node 0 (ground) with a value of 0.1 microfarads. The analysis will be done from node 1 to node 2. The internal resistance of the source does not matter since it only changes the absolute magnitude of the voltage on nodes 1 and 2, not the ratio.
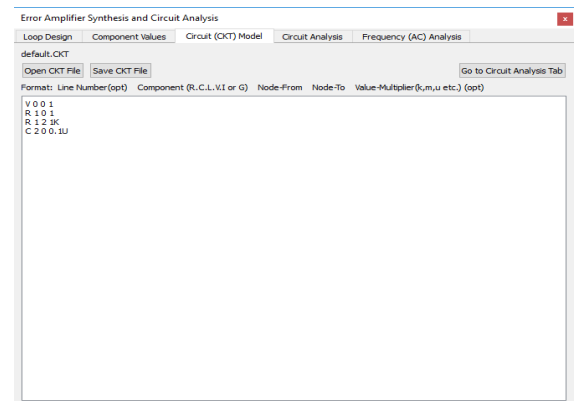
4. After entering the above net list, click the button labeled "Go to Circuit Analysis Tab". Select the desired Max and Min Frequencies (the same ones you used for the test), the nodes to analyze (1 and 2), and Log Sweep at 10 Steps per Decade. When all variables are correctly entered, click the "Compute AC analysis from CKT" button. You will be sent automatically to the "Frequency (AC) Data" tab. An ASCII text result file will be displayed in Venable Standard Format, which is 5 columns: Frequency, Node number, Voltage (volts), Voltage (dB), and Phase (degrees). There will be a row for each node selected at each frequency. There is no fixed limit on the size of the circuit or the number of nodes being
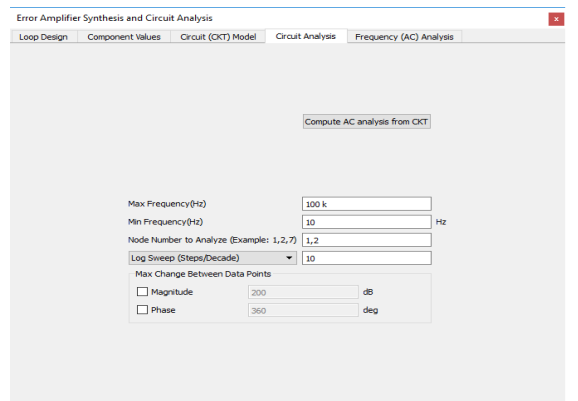
analyzed. The maximum size is dependent on the available memory in your computer.
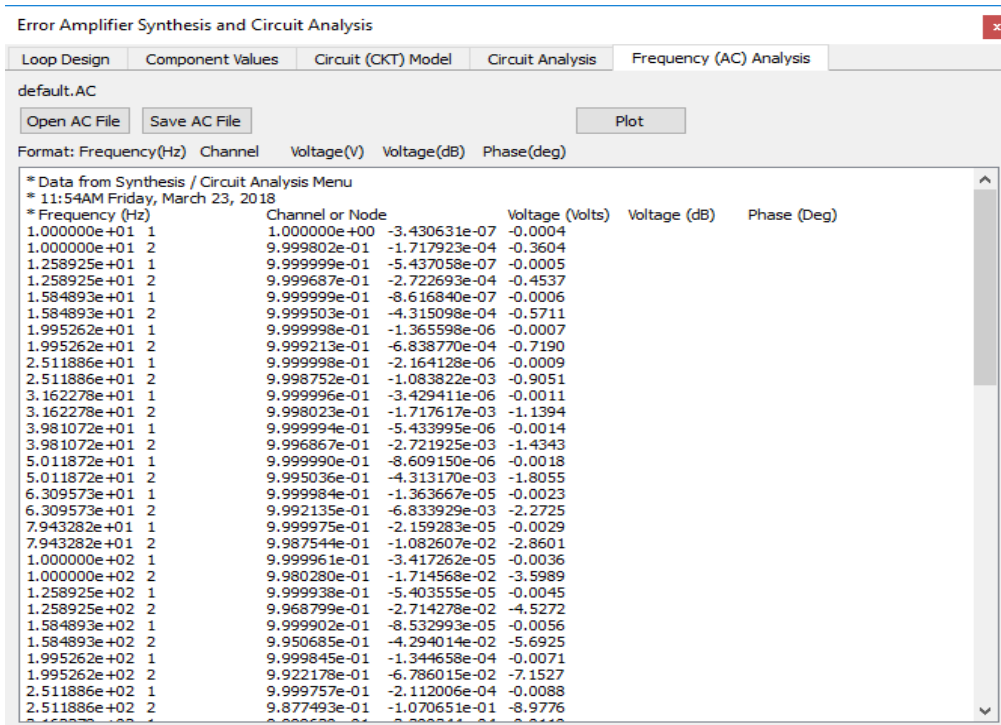
5. Click on the "Plot" button. The Data Set Properties sub window gives you the choice of output node number, input node number, and scale factor. The default settings, Output Node=2, Input Node=1, and Scale Factor=1 are correct for this plot.

6. If you did the test first, both plots will be on the graph. The test data will be labeled 1-Gain and 1-Phase. The model data will be labeled 2-Gain and 2-Phase. If you entered the correct values for the R and C, the two sets of data should virtually overlay each other. You can use the Data Set list sub window on the right hand of the plot window to select and toggle through the data sets displayed on the screen. You can change the properties of a data set by double-clicking to select a data set in the Data Set list window. The number and descriptor of the data set you selected will appear in the Data Set Properties sub window at the bottom of the plot display. You may select another data set and its properties by clicking on another data set in the Data set list. You can document any data set by entering a description in the text window. Other data set properties you may alter are line style, line thickness, scaling, channel ratio, gain or phase color. When you are done click "Save Changes" and "Close". The data can now be documented for printing. You may also wish to display the values of the gain/phase data at a particular frequency with a slide bar. Click on the Slide Bar icon to create a slide bar, then left-click and drag the slide bar to the frequency of
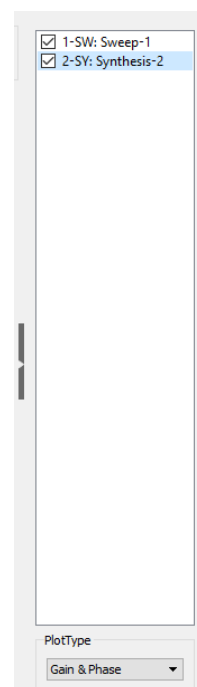
interest. A particular data set can have multiple slide bars if needed. The Slide Bar windows may also be moved anywhere on the chart that is convenient before printing.
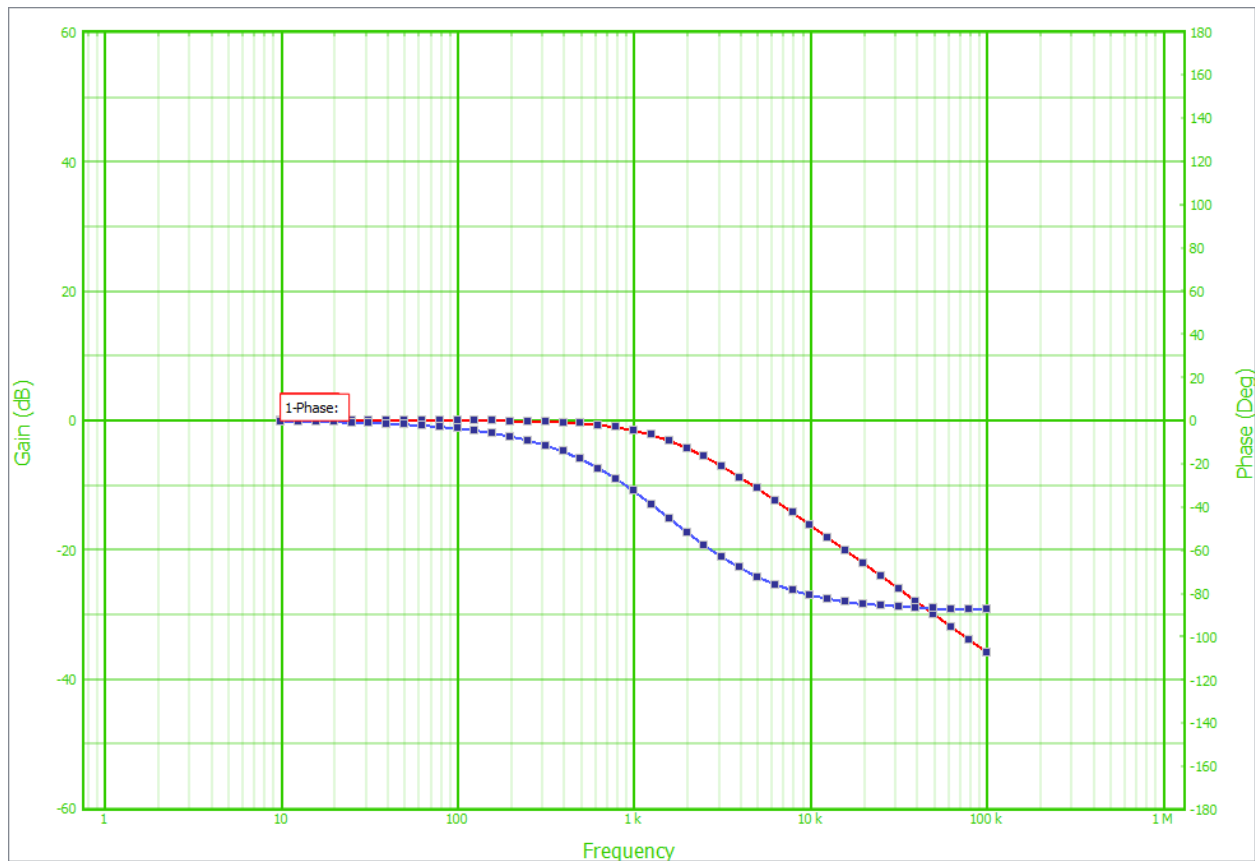
RC Low Pass Filter Model Data Plotted Over Test Data

Toolbars and Drop-Down Menus
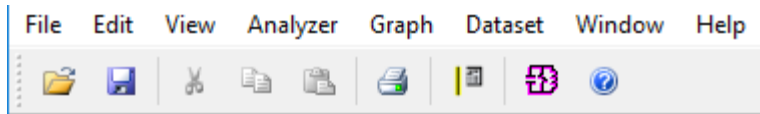
The toolbar is shown above. The first six icons are Windows standard icons, Open (plot file), Save, Cut, Copy, Paste, and Print. The remaining three icons are (left to right):

1) Add Slide Bar
2) Circuit Analysis and Synthesis menu
3) Help

To see the functions available on the drop-down menus (File through Help), click on the Help menu and then click on Venable Help or click the Help icon. You can click on the individual menus when the Help window opens. An index of these is shown below:
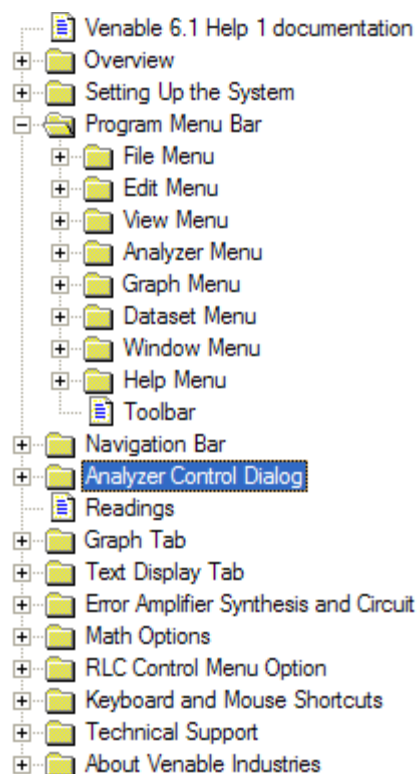
Tools Menu

Venable (Data File) Converter

This menu option will start a data file
conversion tool in a separate window.
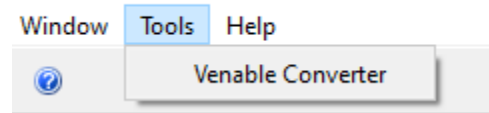
It is a software utility that allows users to convert the *.ven file format between Venable
Stability Analysis™ Software Version 6 and Version 5.x (or earlier). This allows the
interchange of data files between users running different versions or importing existing
data files from older projects.

After starting up the tool, use the File menu option to open an existing *.ven file and also
to save it to another version format.

Keyboard and Mouse Shortcuts

Like any Windows program, you can hold down the Alt key and hit any underlined letter to activate that pull-down menu. Many of the standard Windows Edit Menu <u>control key functions</u> also work, for example:

Ctrl+A	Select All Data Sets
Ctrl+C	Copy Data Set
Ctrl+N	New Plot File
Ctrl+O	Open Plot File
Ctrl+P	Print Plot File
Ctrl+S	Save Plot File
Ctrl+V	Paste Data Set
Ctrl+X	Cut Data Set

The Analyzer Control has <u>keyboard shortcuts</u> associated with it and selecting the parameter with the mouse.

Up or Down arrow key	Increments parameters in the Analyzer Control window according to preset multiplier or integer step sizes.
Page Up or Page Down key	Increments parameters in the Analyzer Control window according to preset integer step sizes.

Besides the keyboard shortcuts, there are <u>mouse short cuts</u>.

1) Left click on any point of any data set to select it.
2) Left click and drag any window or menu to move it or dock it.
3) Left click and drag the side or corner of a window or menu to resize it.
4) Left click and drag a Slide Bar or a Slide Bar value display to move it.
5) Use scroll wheel OR right click and drag the graph to the left of the Gain or right of the Phase axes to change the gain or phase scaling respectively.
6) Left click and drag the graph to the left of the Gain or right of the Phase axes on the Gain or Phase axes to change the gain or phase range displayed.
7) Use scroll wheel OR right click and drag the graph anywhere between Gain and Phase axis to change frequency scaling.
8) Left click and drag the graph anywhere between Gain and Phase axis to change range of frequency values displayed.
9) Use scroll wheel to change modifiable mouse selected parameters according to preset step sizes.
10) The Graph window has special adjustment bars to hide or display the Slide Bar, Data List, and Data Set Properties. Click and drag the adjustment bars in the direction of the arrows to change the size of the graph display.

Measuring a Feedback Loop Transfer Function

The classical way to measure a feedback loop transfer function is to break the loop at some point, terminate the input with the output impedance, terminate the output with the input impedance, drive a small AC signal into the input and measure the ratio of the output to the input. In SPICE analysis, the input is frequently set at 1 volt and the output voltage is then plotted directly as gain since the denominator of the ratio has a constant value of 1. In real life, this measurement approach is virtually impossible since the loop gain is usually very high at low frequency and it is difficult to keep the input stable enough to prevent the output from swinging wildly from limit to limit.
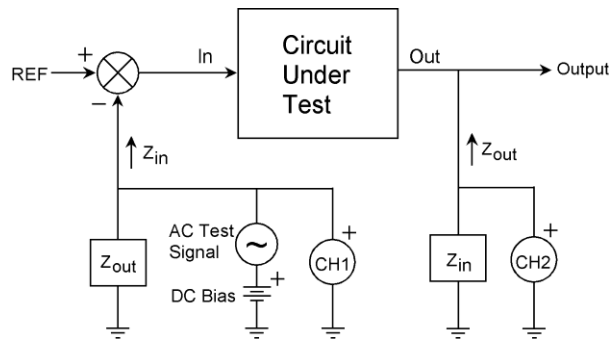
Classical Method
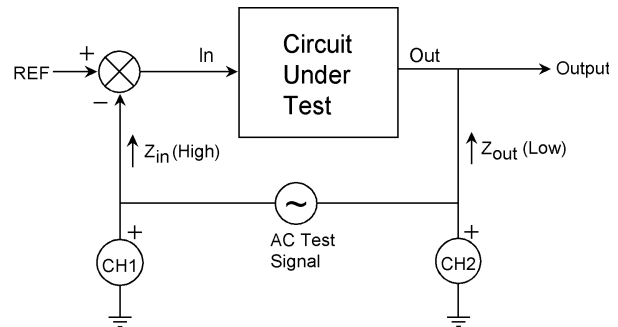
Venable Method

We avoid this difficult measurement situation by finding a place where the loop is confined to a single path (also a requirement in the classical method) and a place where the signal comes from a low-impedance point and drives a high-impedance point. This impedance condition minimizes the error caused by not properly terminating the input and output. We then insert a small resistor into the feedback loop (small compared to the input impedance of the loop). Finally, we connect a floating AC source (the output of a transformer or floating oscillator) across the new resistor and drive it with a sinusoidal voltage source. This converts the resistor into a floating sinusoidal error voltage in series with the feedback loop. This voltage modulates the operating point of the entire circuit.
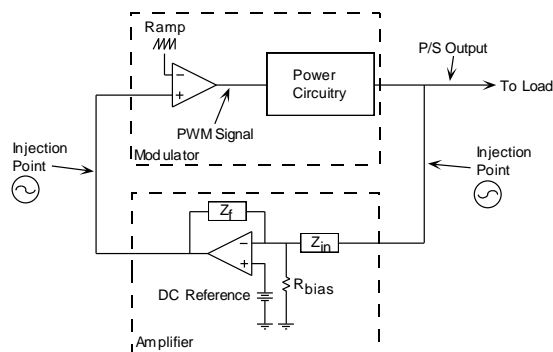
Injection Points
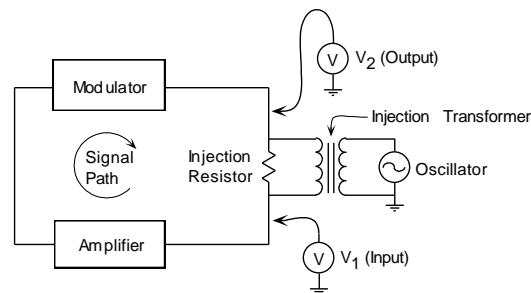
Injection Technique

Once the operating point is modulated, it is easy to measure the voltage to ground from any point in the circuit. The difficult part is that the only signal that matters is the voltage at the frequency of the injected error voltage, which may be a few millivolts in the presence of volts of noise. That is where our superior Frequency Response Analyzer comes in. It uses true Fourier Integral analysis (as opposed to the less-accurate Fast Fourier Transform) to accurately measure amplitude and phase of small signals buried in large amounts of noise. Loop gain is the ratio of the voltage out of the circuit to the voltage into the circuit. The amplitude of the various voltages varies widely with frequency, but the absolute values are not important, only the ratio and relative phase angle.

In the rare case where the impedance ratio of input to output is not sufficiently high, a correction factor can be applied to the data to correct for the impedance condition.

$$A_{actual} = \frac{A_{measured} + \dfrac{Z_{out}}{Z_{in}}}{1 + \dfrac{Z_{out}}{Z_{in}}} \qquad (1)$$

There is also a "dual" of the voltage injection method that is useful when the input impedance is much lower than the output impedance. This involves injecting a current into the loop instead of a voltage and measuring the ratio of the currents on either side of the injection point instead of the voltage. These two test techniques are shown in the figures below, and the current injection technique is used in one of the injection point examples.
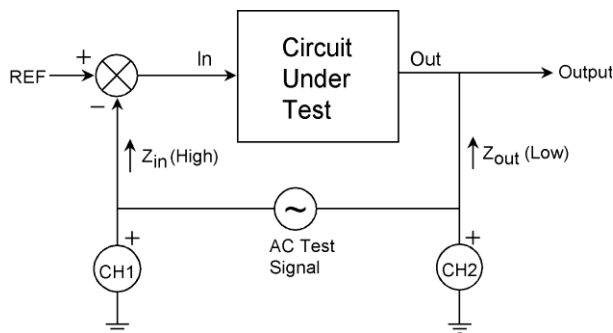
Voltage Mode Injection
Use when Zin >> Zout
Gain = CH2/CH1
This is the normal injection method
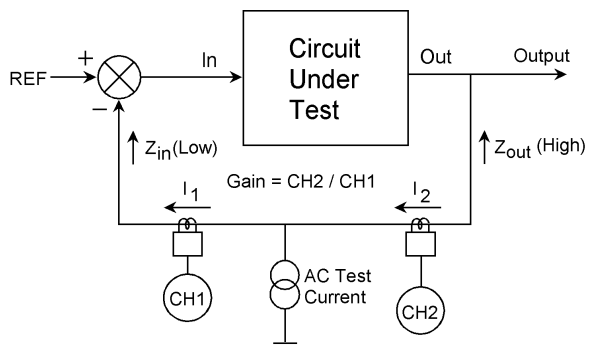
Current Mode Injection
Use when Zin << Zout
Gain = CH2/CH1
"Dual" of normal method

Here is a step-by-step description of how to measure a feedback loop transfer function:

1. Find a place in your circuit where the loop is confined to a single path, comes from a low impedance, and drives a high impedance. In a power supply, the most reliable place that meets these criteria is the point where the resistor or network (labeled Z_i in the figure below) connects to the output of the power supply, typically a large-value electrolytic capacitor.
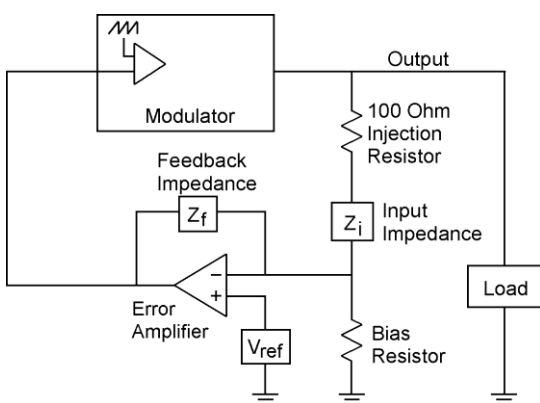
Injection in top of feedback string.
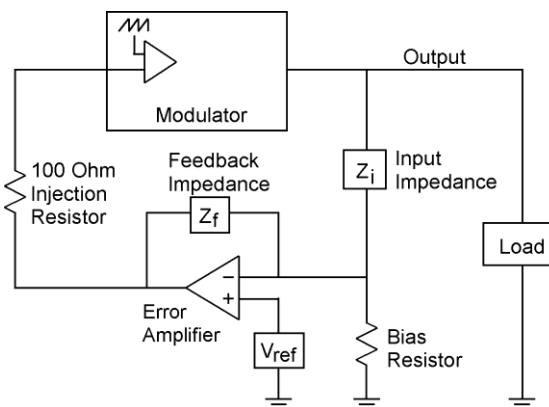
Injection between amplifier and modulator.
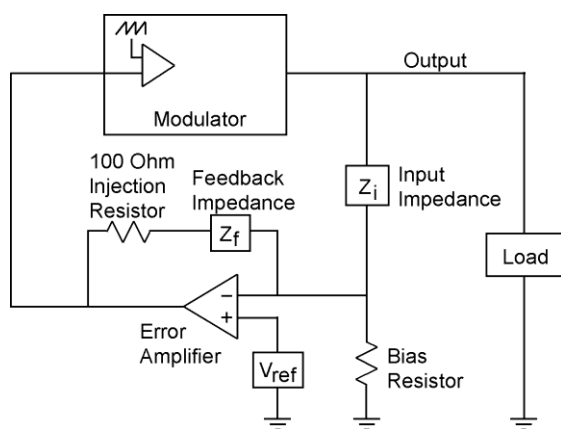
Inject in feedback of error amplifier. Always available but usually noise sensitive. Also works on circuits with TL431's and optocouplers.
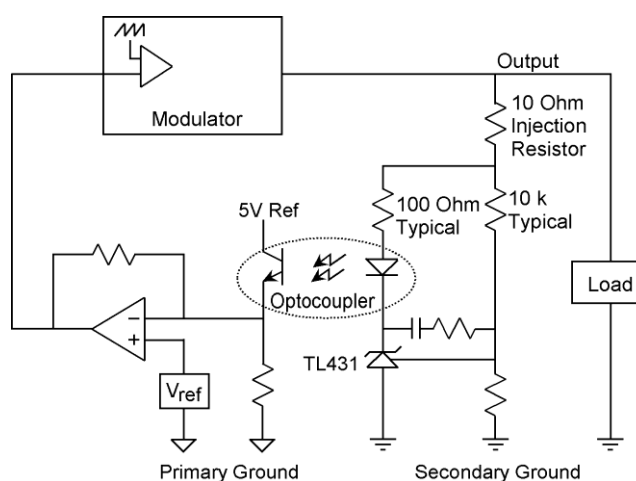
Injection in circuits using TL431. Make sure injection includes both the "fast loop" through the diode and the "slow loop" through the resistive divider string.
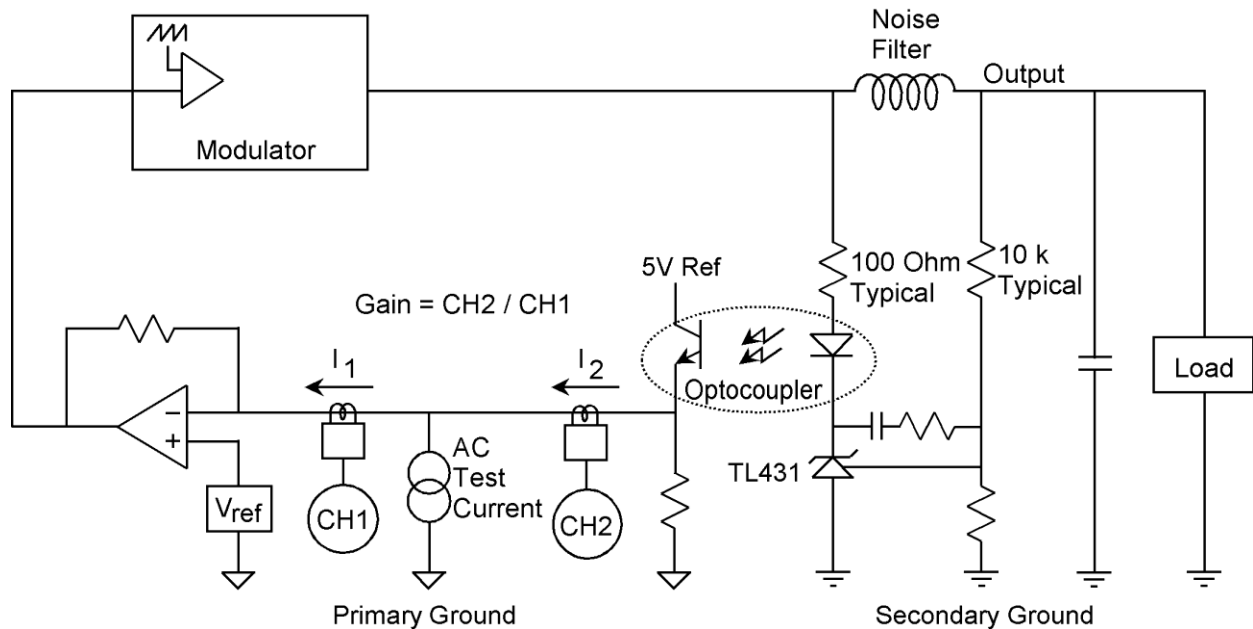
Current injection works in many difficult measurement situations. This method requires two current loops and two current probes but no resistor inserted in the circuit. Gain is determined by the ratio of two currents instead of the ratio of two voltages. A high-value resistor (typically 10k) is used to couple the oscillator output to the test point between the two current probes. Use the DC offset capability of the oscillator to bias the DC level to match the common mode voltage of the error amplifier, usually 2.5 volts, so the DC operating point of the circuit is not disturbed.

2. At the injection point, lift the resistor or network off the printed circuit board and insert a low-value resistor between the network and the output bus. The preferred value of this injection resistor is 100 ohms. Lower resistance values can be used, even down to 1 ohm, but the signal levels and the bandwidth of the injection transformer may suffer. You can judge the effect for yourself by noting the quality of the data, especially the data at frequencies far below or above the loop crossover frequency. On future designs, think about designing the injection resistor in as a permanent part of the circuit. The cost is negligible and it allows you to perform a loop frequency response test any time you want to.

3. Three points are necessary to measure loop gain: the two sides of the injection resistor and signal ground. The connection used to measure signal ground should preferably be very near the ground of the DC reference voltage, typically the ground pin of the PWM control chip. If these three points are not all accessible with Mini-grabber clips, solder test points to the circuit as required.

4. Using three of the BNC-BNC cables provided, connect the Oscillator, Channel 1, and Channel 2 of the Frequency Response Analyzer to the corresponding three BNC connectors on a Bode Box injection transformer.

04/14/20

5. Using the black, brown, and red Banana-Mini-grabber cables provided, connect each banana plug end to the correspondingly colored banana jack on the same Bode Box injection transformer.

6. Connect the black Mini-grabber to signal ground near the ground of the reference voltage. This point must be within rated signal ground to chassis ground voltage of your particular analyzer.

7. Connect the brown Mini-grabber to the point where the injection resistor ties to the error amplifier input resistor or network. This voltage must be within rated input to signal ground voltage of your particular analyzer.

8. Connect the red Mini-grabber to the point where the injection resistor ties to the power supply output bus (or equivalent point in your circuit). This voltage must be within rated input to signal ground voltage of your particular analyzer.
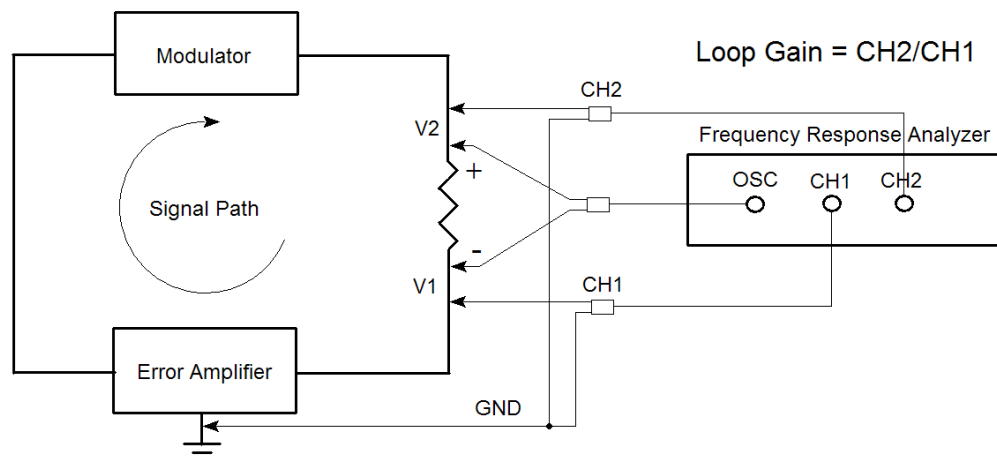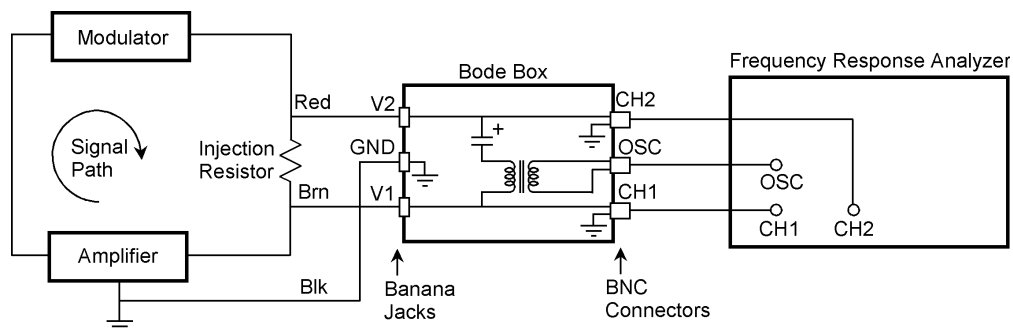
Loop gain measurement setup using a Venable analyzer with a floating oscillator. No transformer is required in this case. Separate BNC-Minigrabber cables are used to inject and measure the loop gain input and output voltages

9. Turn your circuit on and make sure it is operating properly.

10. A highly recommended step is to connect a two-channel oscilloscope to the same signals that Channel 1 and Channel 2 are connected to. This will allow you to monitor the amount of signal injected into the loop and determine if there is any clipping or distortion of the sinusoidal injection signal. The theory behind this measurement requires that the output be continuously and linearly proportional to the input. If this is not true, the results will not be valid.
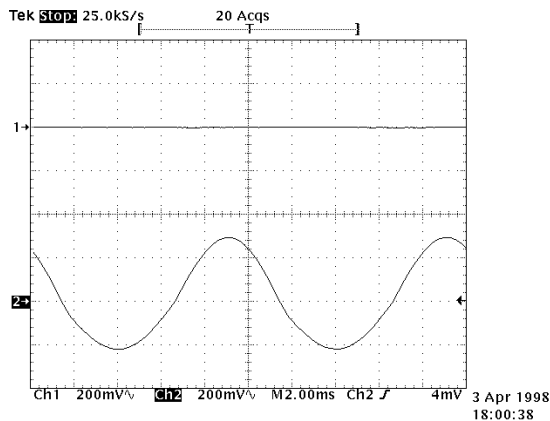
Distorted sine wave indicates overdrive.
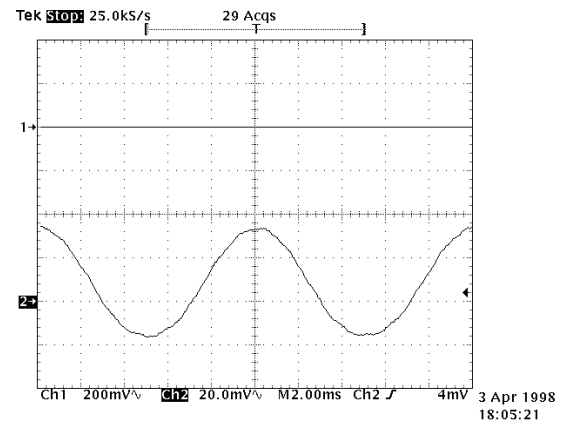
Undistorted sine wave is desired.

11. Open the Venable software. Set the variables in the Analyzer Control window as desired.

Typical values are for 100 kHz switching frequency:

Start Frequency	10 Hz
Stop Frequency	90 kHz (10-20% less than your switching frequency)
Log Sweep	10 Steps/Decade
AC Volts Out	1 V with Bode box or 100 mV with a floating oscillator
DC Volts Out	0
Waveform	Sine
Output Impedance	50 Ohm
Integration	Time
Integration Time	1 Sec
Delay Time	0 Sec
Input Impedance	1 Mohm
Servo	Off
AC Out Step Size	10 dB
Max AC Volts Out	10 Volts peak
Max Gain Change	Off (unchecked)
Max Phase Change	Off (unchecked)
Channel Ratio	CH2/CH1

12. Click on the "Take Data at Start Freq" button. There should be communication with the analyzer, with the data showing up in the Command Display text box. The "Gain" and "Phase" boxes in the Readings window should read reasonable values.

Command Display

```
Sent: ESE,60;MODE,SING;DCOF,0.0;
Sent: BIAS,ON;OUTP,ON;*SRE,1;
Sent: AMPL,4;FREQ,100.0;MEAS;
Sent: READ?;
Received: 1.0000e+02,1.7473e-03,0.0000e+00
Sent: READ?;
Received: 2.6891e-01,1.1790e+02
```

13. If everything looks OK, click the "Run Sweep" button. The system will automatically sweep from the start frequency to the stop frequency at the specified number of steps per decade. You can see the data being presented in real time.
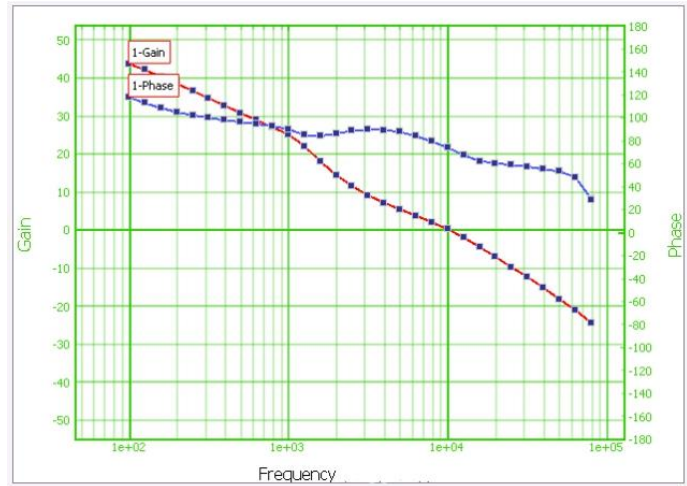
14. The data just taken will automatically be "selected" and each data point will show as a black dot. Below the graph are the data set properties for the selected graph. You can change the ratio of output node number to input node number, scale factor, line type, line thickness or color of either curve. You can also edit the Data Text Description.

Measuring a Modulator (Plant) Transfer Function – Open Loop Method

We use the words "Modulator", "Plant", and "Control-to-output transfer function" synonymously. These terms refer to the gain from the output of the error amplifier to the output of the system. This gain block typically has a fixed low-frequency gain. High frequency gain falls off at a -1 (-20 dB/decade) or -2 (-40 dB/decade) slope depending on the characteristics of the circuit. Because the gain at low frequency (including DC) is fixed, it is possible to use a DC voltage to bias the operating point to achieve a desired system output. By superimposing a small AC voltage on the DC bias voltage, the operating point of the modulator can be varied. The transfer function of this gain block can then be measured by connecting frequency selective voltmeters (the inputs of the Frequency Response Analyzer) to the input and output of the circuit and sweeping the modulation frequency across the desired frequency range. The output of Venable Analyzers are designed to deliver DC and swept frequency AC voltage simultaneously. The inputs are designed to measure voltage at the frequency of the output and reject all other frequencies and DC.

Here is the step-by-step procedure for measuring the control-to-output transfer function:

1. The first step is to set up the circuit to be controlled by the output of the FRA. There are three ways to do this. If the error amplifier is a high output impedance transconductance amplifier, the output can be biased high and the output of the FRA used directly to control the operating point. If the error amplifier has a conventional low impedance output but has an input common-mode range at least equal to the voltage swing needed in the output to control the system, the error amplifier can be wired as a buffer follower. If the error amplifier has a conventional low impedance output and a relatively narrow input common-mode voltage range that does not encompass the entire output voltage swing required, the error amplifier can be wired as a gain stage and there is complete freedom of operating point. If in doubt, this third method will work in any situation. The three methods are shown in the nearby figures.
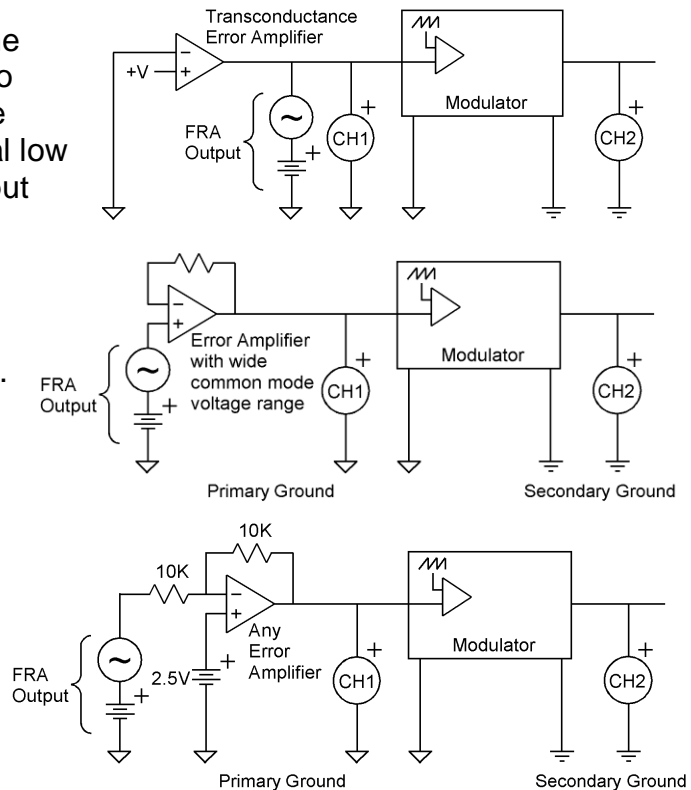
2. The next step is to adjust the DC voltage for the desired output of the system. This can always be done if the system has a fixed gain at low frequency. It may be simpler to use a DC lab power supply or a pot across the +5 V reference voltage from the PWM chip to set the bias voltage initially since they are easy to adjust.
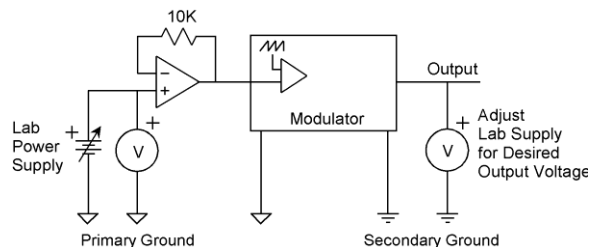
Adjusting operating point with laboratory power supply.
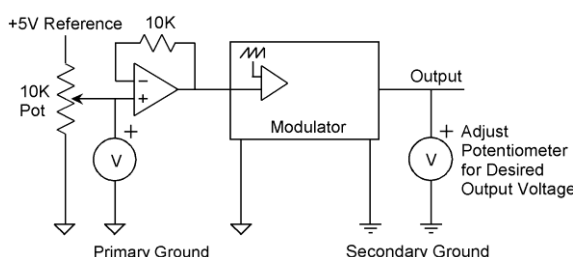
Adjusting operating point with pot across 5V reference of PWM chip.

The DC voltage can then be read with a voltmeter and the FRA output DC voltage pre-set to the approximate operating point. To set the DC output voltage of the FRA, open the Analyzer Control menu and set the "DC Volts Out" to the desired value. Then click the "Turn DC Volts ON" button to set the FRA output to that DC value. Once the approximate operating point is set, it is easy to increment the DC voltage from the FRA to "tweak" the system output to the desired value. The buttons next to the DC voltage setting box increment the DC voltage in 10 mV increments. The DC output voltage is not affected by the sweep.

3. If you haven't already done so, connect Channel 1 of the FRA to the input of the modulator (the error amplifier output) and Channel 2 of the FRA to the output of the modulator (the system output). Use the furnished BNC-Mini-grabber cables unless the system under test is affected by the capacitance of the cables. If it is, use a 10:1 scope probe on both Channel 1 and Channel 2. (See the section on calibrating probes in the advanced applications portion of this manual.)

4. In the Analyzer Control window, set the AC volts to a small number, typically 10 millivolts or less depending on how much you want to modulate the operating point. If you are not sure, start low and raise the voltage slowly while monitoring the amount of modulation you are injecting. When you click the "Take Data at Start Freq" button, the FRA will output that small signal superimposed on the DC voltage. The modulator output will be the modulator input multiplied by the modulator gain, therefore the ratio of output voltage to input voltage is the modulator gain.

5. To get the modulator transfer function, click the "Run Sweep" button in the Analyzer Control window. The measurement system will then automatically sweep the frequency of the AC portion of the FRA output signal from the start frequency to the stop frequency at the specified number of steps per decade and display the gain vs. frequency and phase vs. frequency data on the graph.
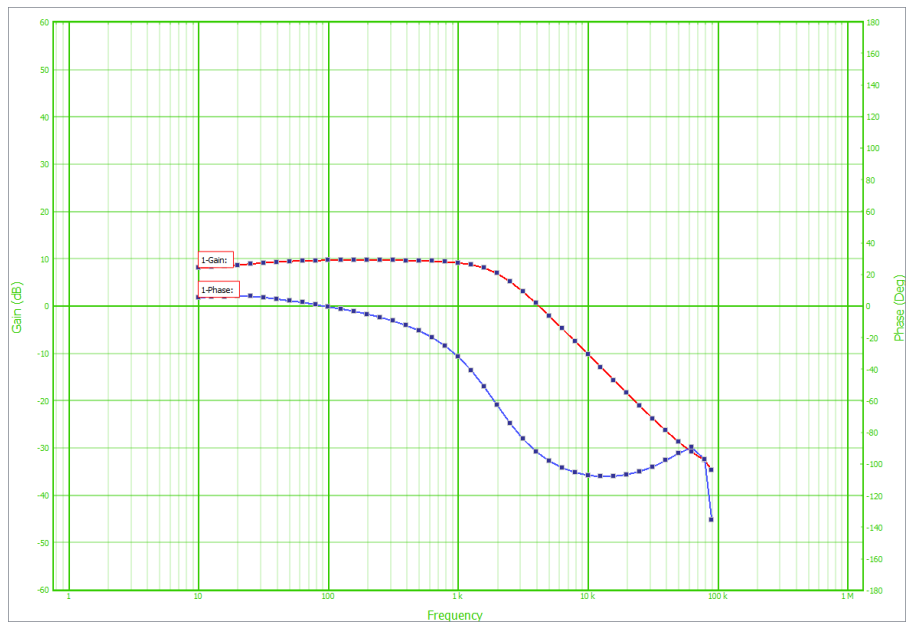
Modulator Transfer Function

Measuring a Modulator Transfer Function – Closed Loop Method

The open loop method described in the previous section is the best method for going directly to the desired system performance without any trial-and-error. Quite often, however, the loop is already closed and the objective is to "fix" the loop rather than design it. In this case, the modulation signal is injected as an error voltage in series with the feedback loop exactly as it is when measuring loop gain (see section on measuring loop gain).

The principal difference between measuring the modulator and measuring the loop is that the simple connection where the injection signal is connected automatically to the FRA inputs through internal connections in the Bode Box injection transformer is not used. The Bode Box injection transformer

is still needed to inject the signal to vary the operating point, unless the analyzer has a floating oscillator. Separate BNC-Minigrabber or scope probe connections are used to measure the modulator input and output voltage, just as in the open loop case.
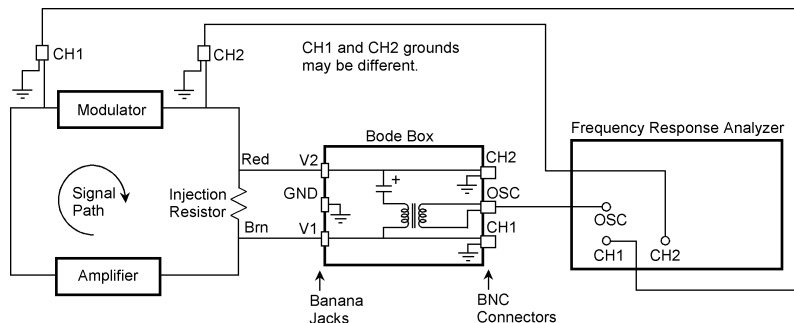
The advantage of measuring the modulator transfer function while the system under test is operating closed loop is that the bias adjustment procedure described in the previous section is not needed. The disadvantage is that the signals are generally noisier and the accuracy is not as good, especially at high frequency.
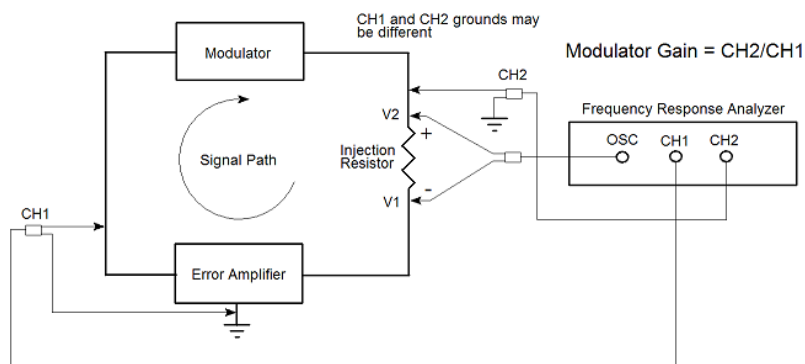
Measurement with a Floating Oscillator and No Transformer

Servo Control Recommended

The Servo Control dynamically changes the AC Volts Out to keep an input channel at a fixed voltage while the analyzer takes data. The Servo Control compares the input voltage on the monitored channel to the AC Volts In and changes the AC Volts Out until they are equal. It changes the AC Volts Out by a maximum of AC Out Step Size. This feature is helpful in preventing the analyzer oscillator from overdriving the circuit past Max AC Volts Out during a sweep. The recommendation is to set Servo on the output channel (typically channel 2) at 20 mV to 30 mV and Max AC Volts Out to your circuit limits below the maximum output of 10 Vpk.

Design of a Feedback Amplifier from the Modulator Transfer Function

Once you have measured (or modeled) the modulator transfer function, you can have the Venable System software design the compensation amplifier for your desired loop. Here are the steps:

1. With the modulator (also called plant or control-to-output) transfer function displayed as a gain-phase graph and selected, click the Slide Bar icon or right-click anywhere on the open graph and click "Add Slide Bar". Note that data is "selected" by clicking anywhere on either curve of the data set or pressing the right- or left-arrow keys to toggle through the various data sets displayed. Also note that the Slide Bar option is not available if more than one data set is selected since the Slide Bar is keyed to a particular data set.
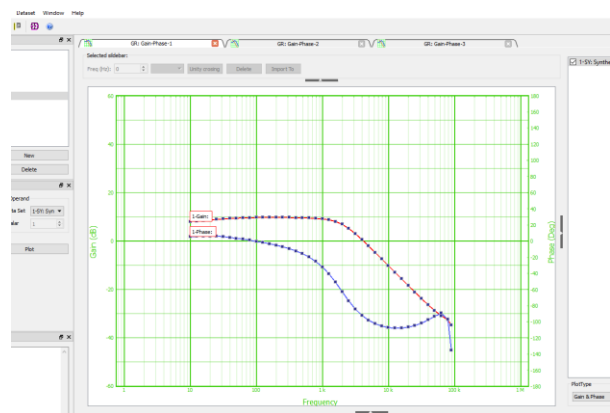
2. When the Slide Bar is added to the modulator transfer function data set, click and hold it with the left mouse button then move it to the desired loop bandwidth. In choosing loop bandwidth, remember that the response time of a system to a transient is approximately the reciprocal of the loop bandwidth. Also remember that the theoretical bandwidth limit of a switching or sampled data system is half the switching or sampled data rate.
As a practical matter, even though it is theoretically possible to cross the feedback loop over at half the switching or sampling frequency, a more achievable number will be in the range of 10-20% of the switching or sampling frequency.
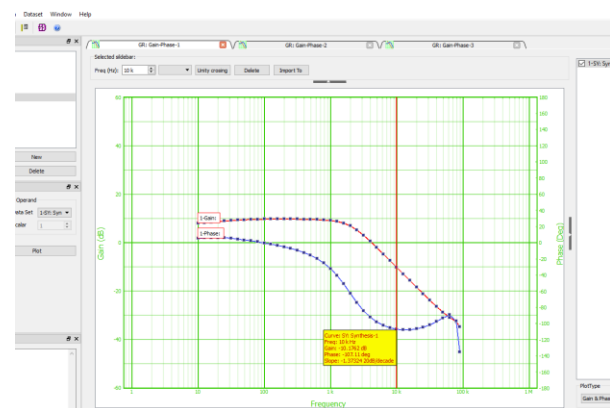
3. After choosing the desired loop bandwidth and moving the Slide Bar to that frequency, note that the text box associated with the slide bar has a button labeled "Import To". Click that button. This will open the "Error Amplifier Synthesis and Circuit Analysis Menu" window and transfer the loop crossover frequency to that window together with the modulator gain and phase measurements at that frequency.

4. In the Error Amplifier Synthesis and Circuit Analysis Menu window there are a number of choices to make:

a) Phase margin – Choose a value between 45 and 72 degrees. We recommend 60 degrees as a good starting point.

b) Maximum circuit complexity – Type 1 is a simple integrator, type 2 has a single zero-pole pair, and type 3 has two zero-pole pair. If the modulator transfer function is flat at the chosen crossover frequency, choose type 1. If the modulator transfer
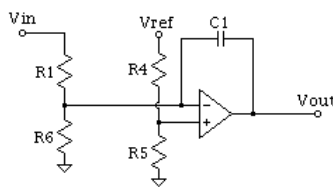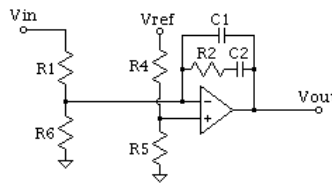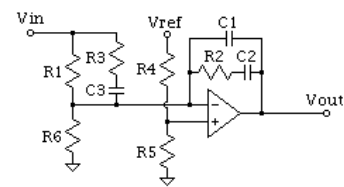
Type 1   Type 2   Type 3

function is falling at a -20 dB/decade slope at the chosen crossover frequency, choose type 2. If the modulator transfer function is falling at a -40 dB/decade slope at the chosen crossover frequency, choose type 3. An error message box on the next (Component Values) tab will tell you if you chose an unachievable combination.

c) "Circuit is Inverting" check box – Uncheck this box if the modulator inverts (output is 180 degrees out of phase with the input) at low frequency. Leave it checked if the modulator output is in phase with the input at low frequency. If the gain is flat at low frequency (it should be), the circuit will be either inverting or non-inverting. It will not have some in-between phase value if the test was performed correctly.
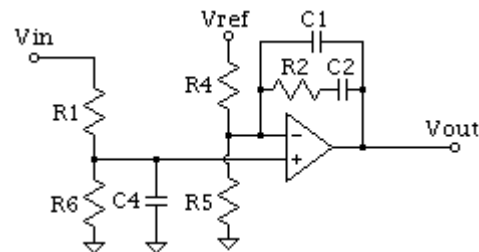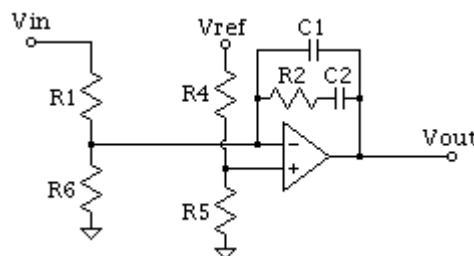
Inverting amp for non-inverting modulator   Non-Inverting amp for inverting modulator

d) Amplifier Topology – Regular is a conventional low output impedance amplifier where the feedback compensation components are connected from error amplifier output to inverting input. Transconductance is a high output impedance amplifier where the feedback compensation components are connected from error amplifier output to signal ground.
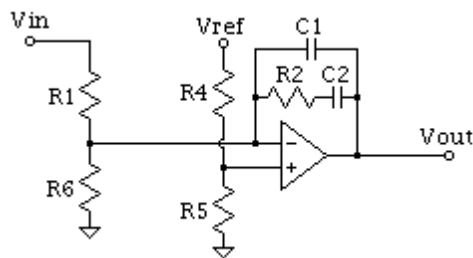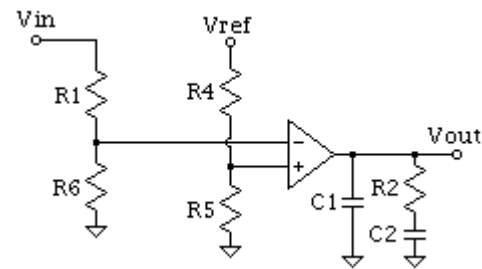
Regular   Transconductance

e) Input Resistor (R1) – this is the value of the resistor, which connects from the modulator output to the error amplifier input. All feedback compensation components scale from this resistor. If you want some other component in the compensation network to have a particular value instead, choose a round number like 1k for R1 and note the value of the component you want controlled. Then change the value of R1 to make the other component come out the value you want. All resistor values will be directly proportional to R1. All capacitor values will be inversely proportional to R1.

f) Input Voltage – this is the output voltage of the modulator or plant (which becomes the input voltage of the compensation amplifier).

g) Reference Voltage – this is the value of the reference voltage used to control the output of the modulator. It could be a temperature-compensated zener diode, the reference pin of a control integrated circuit, or an external reference or control voltage.

h) Common Mode Voltage – this is the voltage at the inputs of the compensation amplifier. Both the inverting and non-inverting inputs of the amplifier will be at the same voltage if the circuit is operating properly. The synthesis program will determine the proper value of the resistive divider string from Reference Voltage to the amplifier inputs so that the DC resistance driving each amplifier input is the same. This is necessary for best amplifier performance.

i) The only other entry is the error amplifier internal characteristics. Gain and bandwidth come from the data sheet. If you do not know the amplifier output impedance, use 1k. For transconductance amplifiers the gain, bandwidth, and transconductance are specified on the data sheet.

5. After you have entered in all the design values and choices, if you want to save the values for later use click the "Save Design File" button and you will be prompted for a file name and location. If not, go ahead and click the "Compute Component Values" button. This will send you to the next (Component Values) tab with the error amplifier compensation component values calculated already.

6. At the Component Values tab, first check the "Error messages" text box to see if anything needs fixing. If not, choose the preferred tolerances for components in your company, then click the "Round off Component Values" button to round off the values to the nearest standard value for the specified tolerance. We do not use the tolerance in calculations, so you can enter a larger tolerance if you want more standard values even though you plan to use a closer tolerance part in the final design. For example, you may prefer to have the system choose a 1k resistor instead of 910 ohms, so you can choose 10% or 20% tolerance instead of 5% which might give you a value like 910 ohms. The information boxes in the lower right quadrant of the window tell you where the poles and zeros of the amplifier transfer function will be. You can also change any particular component value if a similar value is more readily available and the new pole-zero locations will be calculated and displayed so you can judge the impact of the change on overall loop performance. Once the component values satisfy your criteria, click on the "Make CKT model from component values" button.
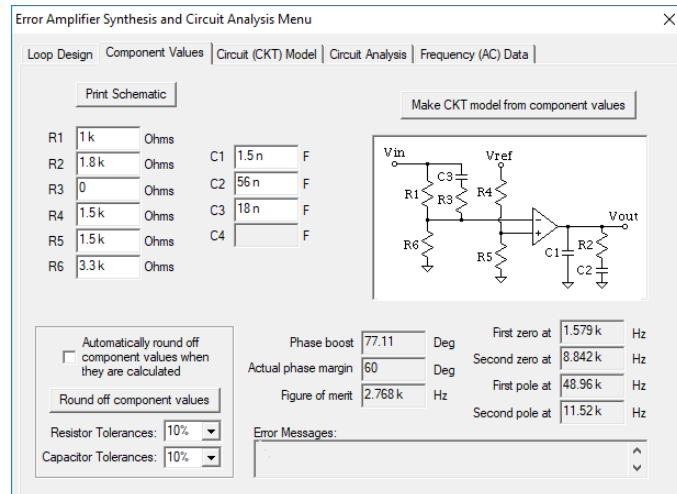
7. This will send you to the "Circuit (CKT) Model" tab and display a SPICE-like net list of the model of the compensation amplifier. See the schematics and models below for reference. This net list can be edited. Components can be added or removed or their types, values or connections changed. The one thing that cannot be done is that a node cannot be removed (skipped in the node number sequence). Node numbers must be sequential starting with zero (ground). If you change a circuit enough to remove a node, simply connect a resistor from the node to ground. It will not have any effect on the circuit and the analysis will still run properly. When you are happy with the net list, click the "Go to Circuit Analysis Tab" button and you will be sent to the next (Circuit Analysis) tab.
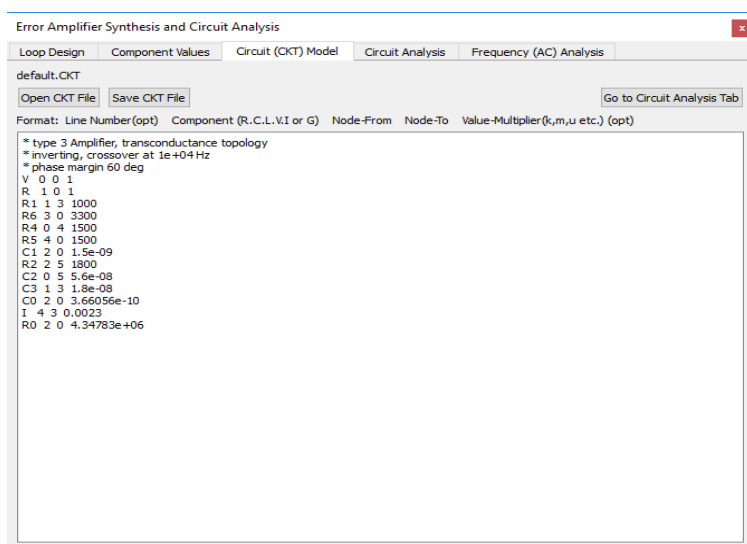
The figures below represent the amplifier schematics and models. The node numbers and component reference designators are correct. The schematics and models differ slightly for transconductance amplifiers, non-inverting amplifiers, and amplifiers where the input voltage and reference voltage have different polarities.
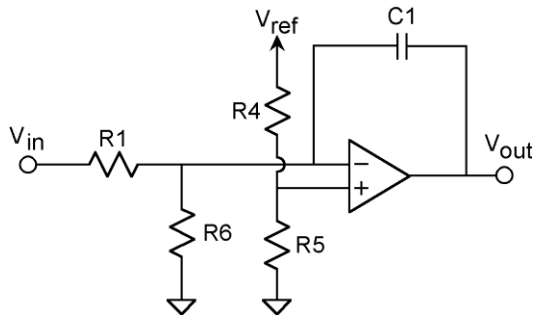
Type 1 Amplifier Schematic
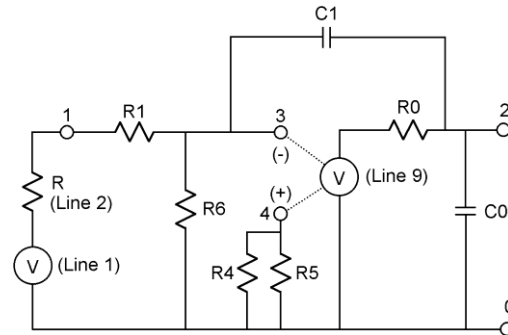
Type 1 Amplifier SPICE Model
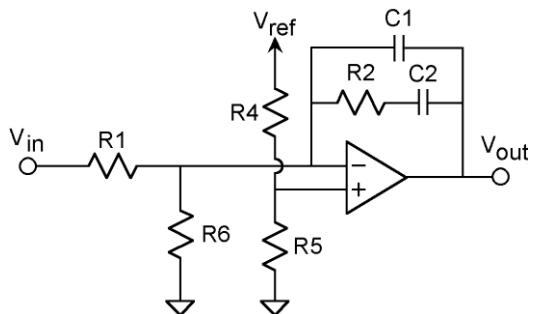
Type 2 Amplifier Schematic
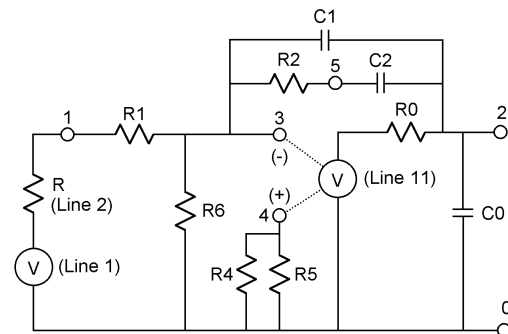
Type 2 Amplifier SPICE Model
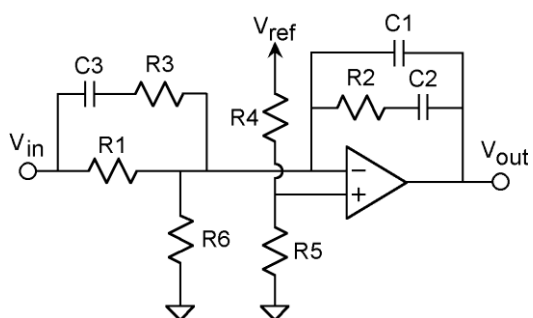
Type 3 Amplifier Schematic
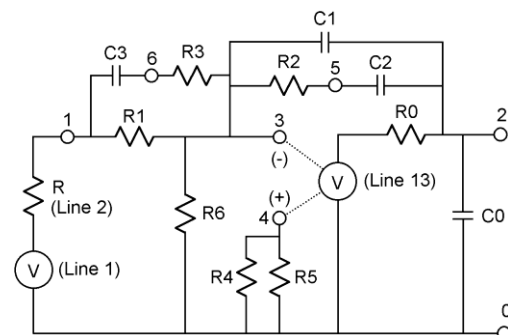
Type 3 Amplifier SPICE Model

8. The Circuit Analysis tab is where you set up the parameters of the AC analysis of the net list. Choose the maximum and minimum frequency for the analysis sweep, the node numbers you want to save data for (you can save all of them if you want to), whether you want a log or linear sweep and what the resolution of the sweep is, and finally whether or not you want to activate the feature we formerly called "Max Delta Gain/Max Delta Phase". This feature measures the change in gain and phase from one analysis point to the next, and if the gain or phase change on any node is greater than the amount specified, will cut the frequency step size in half and try again. This can happen a maximum of 5 times (steps/decade x 32) up to a maximum resolution of 2000 steps/decade.

9. When the parameters of the analysis are set to your requirements, click the button labeled "Compute AC analysis from CKT". This will take you to the final tab, "Frequency (AC) Data". A brief description of the origin of the data will be at the top of the text display in lines starting with * (asterisk). Following the comment lines, the data will be displayed in Venable Standard Format which is 5 columns: Frequency, Node number, Voltage (volts), Voltage (dB), and Phase (degrees). There will be a row for each node selected at each frequency.

10. From the "Frequency (AC) Data" tab, click the "Import into Graph" button. If you have followed the steps up to here, a gain-phase graph of the modulator transfer function is already displayed and the data set properties sub-window, at the bottom of the plot, will display the channel ratio and scale factor. The default values are the ratio of CH2/CH1, and 1. You can also change the line style, line width, and line color of any of the selected plotted data here.

11. After closing the Synthesis menu, you will see the graph with the amplifier transfer function gain and phase superimposed on the original modulator transfer function. At the desired loop crossover frequency, the amplifier gain should be as far above (or below) the 0 dB gain axis as the modulator gain is below (or above).
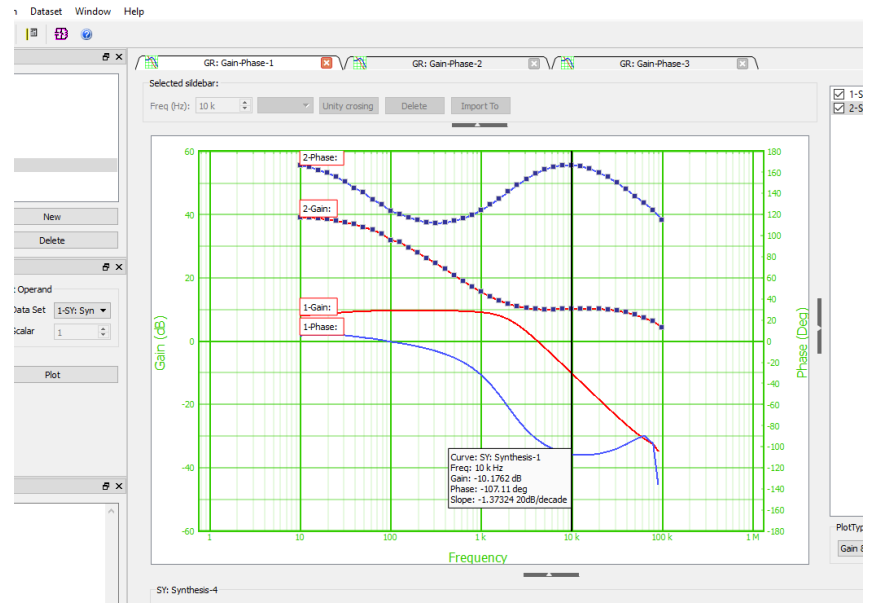
Calculation of the Predicted Overall Feedback Loop Transfer Function

Before you install the various resistors and capacitors of the compensation network in the circuit, you would probably like to know what the overall feedback loop transfer function will look like. After all, you may want to do some tweaking before you solder everything in. Predicting the loop gain is easy. Here are the steps:

1. Display both the modulator transfer function and the amplifier transfer function on a gain-phase graph. If you followed the steps in the previous sections, they are already displayed. Note the data set number of each of the two data sets (the modulator gain and phase curves are a data set and the amplifier gain and phase curves are another data set).

2. If it isn't already open, select the Math Options for Gain-Phase Data in the View program menu. In the "Math Options for Gain-Phase Data" window, set the left operand to the number corresponding to one data set and the right operand to the number corresponding to the other data set. This will usually be 1 and 2 if only two data sets are displayed. Select the "Multiply" action, and then click the button called "Plot".

3. When you click the "Plot" button, the loop transfer function will appear on the graph as an additional data set. It will be "selected" and you will be able to see the individual data points. The loop gain should cross the 0 dB line at approximately the frequency you chose as the desired loop frequency and the phase margin should be approximately the phase margin you asked for. The differences between actual and desired values are caused by rounding off the component values and by the limitations of the op-amp selected. If you don't round off the component values and you do set the gain and bandwidth of the op-amp at high values, the actual and desired values will be identical. In real life, however, you have to use available components and real op-amps and the predicted results will be very close to the actual measurements.

4. If you want to document the results, select (click on) only the final gain and phase plots and add slide bars by clicking on the slide bar icon (the one with the yellow vertical line). Slide bars are keyed to data sets so the slide bar icon is available only when one data set is selected. You can have as many slide bars as you want, but two is usually a nice number. You can put one slide bar on gain crossover and the other on phase crossover. That will give you numerical readouts of the loop bandwidth, phase margin, zero phase bandwidth, and gain margin. As a final

touch, you can add a text box by clicking on the text box icon (with the "A" on it). A brief description of the source of the data and the time and date it was created are entered automatically in the box. You can edit or add to this information to document the model and serial number, test conditions, values of the various components, or anything else you think will be of value when you or someone else looks at the plot.
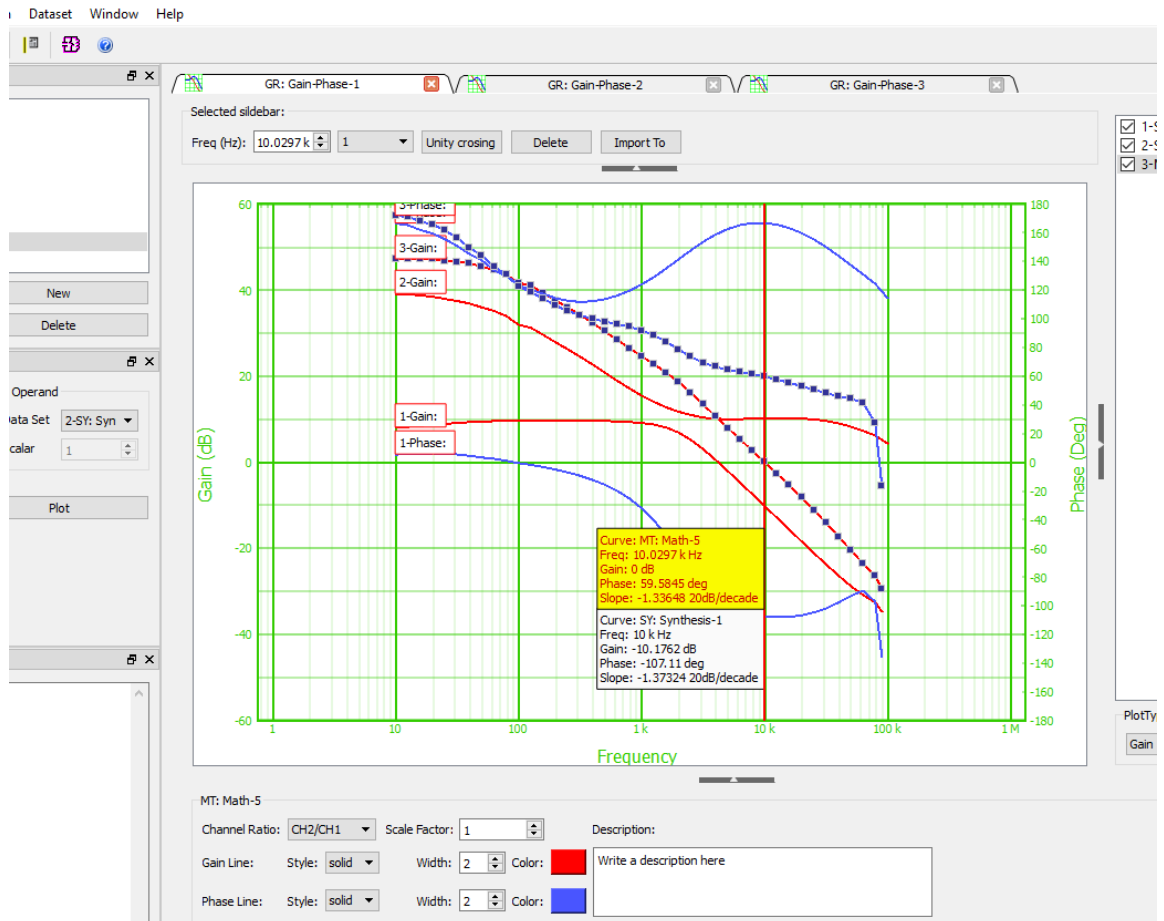
Display Documentation Features

This section details the documentation feature that can be added to the graph displays. The Slide Bar and its associated window are useful for tracking and displaying data values at a specific frequency.
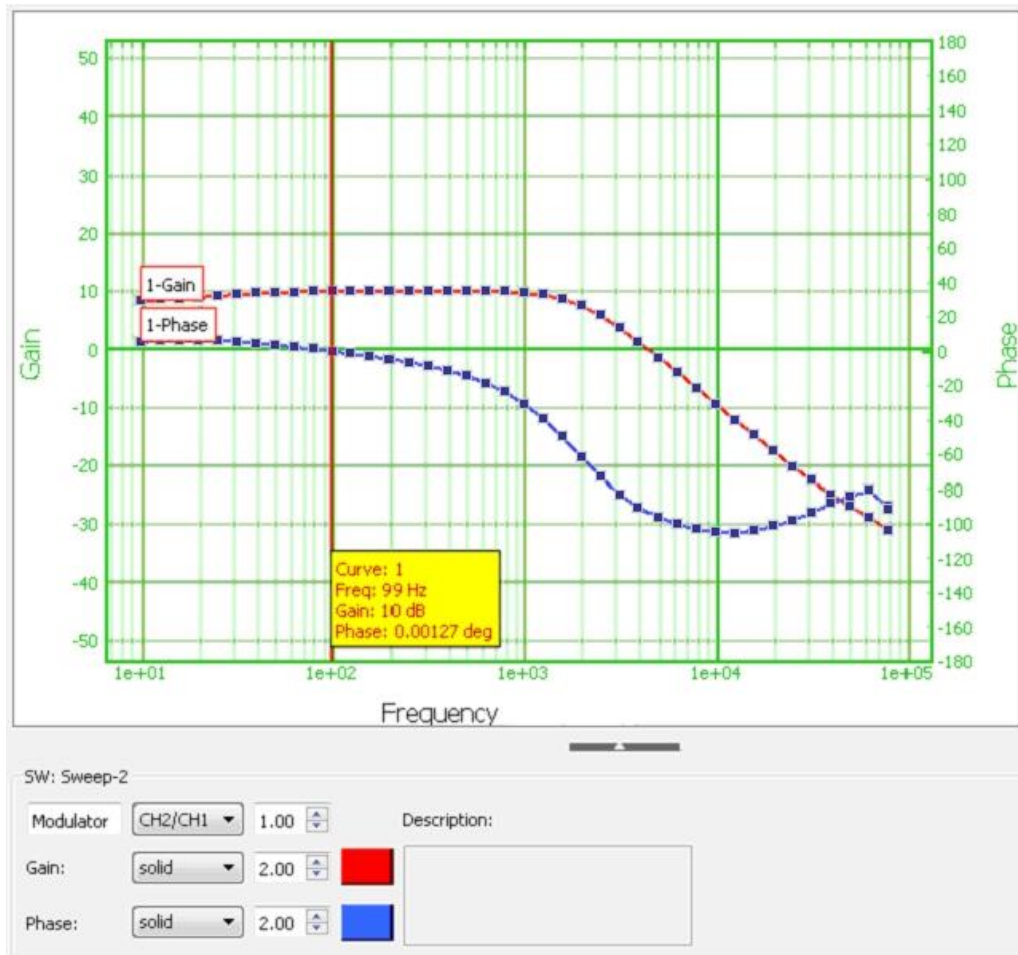
The screen shot above shows the display documentation features for Venable System version 6 software plots.

Slide Bar Tag

Add Slide Bar

This function is located under the Graph program menu. The slide bar is useful for tracking and displaying data values at a specific frequency, represented by a vertical line and a description box. The Add Slide Bar command can only be run when a single data set is selected. More than one slide bar may be opened at one time, either from the same or different data sets. Add Slide Bar is not available in a text display window.

The bar can be dragged using the mouse pointer, and the box will contain the data at the selected frequency. The box can be dragged up or down along the slide bar using the mouse.

The user can click on the Get Unity Crossing button to find the 0 dB crossing point.

Exporting Data from the Venable Software

The Venable software uses tab delimited ASCII text when exporting or importing data from the Graph tab or Dataset Text Display tab. The Venable software saves data in this form when you select, cut or copy, and paste data to the clipboard or save it to a text file. Data can be saved in comma-separated values format by selecting the data and then selecting Export to CSV in the Dataset program menu. Selected data also can be saved as a Matlab .mat file using Export to Matlab in the Dataset program menu. Data in this form can be easily exported to or imported from Excel, Matlab, or Mathcad applications. Screenshots of Graph tabs can be saved to .jpg, .png, or .svg format using the Export to JPG, Export to PNG, and Export to SVG menu selections in the Graph program menu.

Venable Data Format in the Dataset Text Display

Venable data can be exported to the clipboard in three different text formats that can be set in the Dataset Text Display tab. The Voltage format is the multi-column raw data acquired from each analyzer channel. The number of columns depends on the number of analyzer channels   The Voltage format also allows users to examine the raw measurements and do their own calculations on the data. Voltage format can be set to either Volts or dB. The Frequency Magnitude-Phase format is the 3-column text representation of a particular data set plotted on a Venable software graph tab. The majority of users will find this the most useful format for exporting or importing data. The plotted data set consists of a channel ratio multiplied by a scale factor. The channel ratio and scale factor settings for each plotted data set can be changed and are also located in the Data Set Properties for each plot. Changing the channel ratio or scale factor in the Dataset Text Display tab does not change the channel ratio or scale factor in the Graph tab. That has to be changed in the Data Set Properties for the data set.

1. To view the three different data formats, open up a Venable graph tab containing plotted data sets.

2. You can observe the data in the different text formats by going to the Navigation Bar in the Venable application and double clicking on one of the datasets listed.

3. A Dataset Text Display tab containing the data will open. There are three radio buttons that can be selected to set the data in either voltage (volts) mode, voltage (dB) mode or Ratio mode. There are sample screenshots of the data on the next page.

Voltage Mode:

	Freq(Hz)	Ch1(volt)	Ch1(deg)	Ch2(volt)	Ch2(deg)
1	10.0	0.025623	0.0	0.066828	5.4514
2	12.59	0.025152	0.0	0.067051	6.1199
3	15.85	0.024605	0.0	0.067207	6.4354
4	19.95	0.023983	0.0	0.067314	6.491
5	25.12	0.02339	0.0	0.067529	6.1999
6	31.62	0.022834	0.0	0.067576	5.5861
7	39.81	0.022344	0.0	0.067614	4.5837
8	50.12	0.021984	0.0	0.067623	3.5197
9	63.1	0.021703	0.0	0.067629	2.4497
10	79.43	0.021536	0.0	0.067624	1.1653
11	100.0	0.021354	0.0	0.067773	-0.051849
12	125.9	0.021516	0.0	0.067773	-1.6581
13	158.5	0.021319	0.0	0.067774	-3.079
14	199.5	0.021325	0.0	0.067764	-4.6062

Ratio Mode:

	Freq(Hz)	CH1/CH2(dB)	CH1/CH2(deg)
1	10.0	-8.32656974702	-5.4514
2	12.59	-8.51665469907	-6.1199
3	15.85	-8.72782280672	-6.4354
4	19.95	-8.96403781908	-6.491
5	25.12	-9.20920193459	-6.1999
6	31.62	-9.42420968912	-5.5861
7	39.81	-9.61751413161	-4.5837
8	50.12	-9.75975436692	-3.5197
9	63.1	-9.87226390685	-2.4497
10	79.43	-9.93871626847	-1.1653
11	100.0	-10.0315494381	0.051849
12	125.9	-9.96590349363	1.6581
13	158.5	-10.0459257746	3.079
14	199.5	-10.0421998789	4.6062

Exporting Venable from the Data Text Display to the Clipboard

The Venable plot data without the data header is exported to the clipboard as tab delimited ASCII text. Here are the steps you should follow to export data from the Venable software to the clipboard and to paste the data to another application:

1. With the Dataset Text Display tab open as described in the previous section, select all the data text to be exported by left clicking on the upper left corner of the spreadsheet or opening the Edit program menu and select Select All. You can also left-click on a cell and drag to select all the data or a portion of the data.

2. When the data to be exported is selected, go to the Edit program menu and select Copy or use the Ctrl-C shortcut keys.

3. Open up a text editor, like Notepad or WordPad, and directly paste the data into it. Make sure to save the data as a text file in WordPad.

4. Or open up a spreadsheet application like MS Excel and directly paste the data into the workbook.

Importing Venable data into Excel

To import data into Excel you can directly paste any data copied on to the Clipboard from the Venable software directly into an Excel worksheet or you can save a selected data set to a .csv file by going to the Dataset program menu and selecting Export to CSV. You can open up any tab delimited text files that have been saved. When opening a delimited text file with Excel, the Text Import Wizard panel will open up. Choose tab delimited option and click on Finish to import the file.

Importing Venable data into Mathcad

Data can be imported into Mathcad using several methods:

1. The user can go to the Mathcad Insert program menu, select the Component menu item, and then select Input Table. Type a variable name into the placeholder for the input table. Right click on the input table and select Import from the pop up menu. Browse for a data text file with the Venable text header removed and click Open. This method reads the data from the data file only once when the data is imported instead of each time the work sheet is calculated.

2. Data can also be imported by going to the Insert pull-down menu, selecting the Component menu item and then selecting File Read or Write. The File Read or Write Wizard should appear. Choose Read from a file, Text Files or Matlab for the File Format, and browse for the data file and click on Finish. Type a variable name into the placeholder in the Read/Write component icon

to which the data from the file will be assigned. Type the variable name and '='to display the data in an input table. Mathcad re-reads the data from the file each time the work sheet is calculated.

3.   Data can also be imported using the READPRN function. The proper syntax is to put the file path inside quotes inside the function parentheses as shown below:

M:= READPRN("C:\Program Files\Venable Inc\Venable System\mod.dat")

Type the variable name and '=' to display the data in an input table.

Importing Venable data into Matlab

To import data into Matlab you can save a selected data set to a .mat file by going to the Dataset program menu and selecting Export to Matlab. You can open this file in Matlab and the data will be loaded into your workspace. You can also import Matlab files into Mathcad. See the previous section about exporting data to Mathcad.

Venable Data Format in the Graph Tab

Venable data can be exported or imported from the Graph tab in the 5-column voltage format. This format is compatible with all the older versions of the Venable software and the data output of the Circuit Analysis menu. The Voltage format is the raw data acquired from each analyzer channel. The Voltage format also allows users to examine the raw measurements and do their own calculations on the data. Voltage format shows magnitude in both Volts and dB. The Venable plot data includes data header set that consists of the analyzer model, date and time, channel ratio, scale factor and column headings.

```
*Data gathered by the Venable 350 System
*04:59PM Sunday, July 22, 2012
*Gain/Phase: Output Node=2, Input Node=1, Scale Factor=1
*Frequency (Hz)   Channel or Node   Voltage (Volts)   Voltage (dB)   Phase (Deg)
100              1                 0.00315526        -50.0193       3.33308
100              2                 1.42425e-005      -96.9283       107.771
100              3                 5.91096e-007      -124.567       168.69
112.2            1                 0.00314995        -50.0339       3.53305
112.2            2                 1.41581e-005      -96.9799       110.364
112.2            3                 5.34378e-007      -125.443       130.601
125.9            1                 0.00315427        -50.022        3.7909
125.9            2                 1.57248e-005      -96.0683       113.92
125.9            3                 3.88818e-007      -128.205       -63.4349
141.3            1                 0.00315708        -50.0143       4.19767
141.3            2                 1.89814e-005      -94.4334       106.498
141.3            3                 2.08983e-007      -133.598       -56.3101
158.5            1                 0.00316046        -50.005        4.62527
158.5            2                 2.22e-005         -93.073        104.98
158.5            3                 2.95548e-007      -130.587       -168.69
177.8            1                 0.00316015        -50.0059       5.11935
177.8            2                 2.35869e-005      -92.5466       101.338
```

Exporting Venable Data from the Graph Tab to the Clipboard

The Venable plot data including the data header is exported to the clipboard as tab delimited ASCII text. Here are the steps you should follow to export data from the Venable software to the clipboard and to paste the data to another application:

1. With the Sweep window displayed as described in the previous section, select the data set to be exported from the plot by left clicking on the data (Black squares appear on the data set when it's selected).
2. When the plot to be exported is selected, go to the Edit program menu or right click on the graph to open the graph Edit menu and select Copy or use the Ctrl-C shortcut keys.
3. Open up a new graph tab or select another graph and directly paste the data into it.
4. Or paste the data into another open Venable application, including older versions of the software. Conversely, you can import data another open Venable application. For version 5 and earlier software, make sure the Voltage Format for Cut/Copy is checked in the Edit program menu.

Exporting a Venable Plot as a Graphics File

The Venable can save plots in JPEG, PNG, or SVG graphics format, which can be read by any web browser, or easily pasted into Word. The functions for exporting graphics files are all located under the Graph program menu.

Export Graph

Export is located under the Graph program menu. Graphs can be exported in several formats, including JPEG, PNG and SVG. It copies the active graph including slide bars to the clipboard in the chosen format using the default compression. The plot can then be inserted into other programs such as Word or PowerPoint for reports or presentation. Another method is to use Alt-Print Screen to capture the active graph on to the clipboard as a Windows bitmap. In this case, you will have to use a graphics file editor to remove the window portion from around the graph.

Description and Uses of Accessories

Part number	Description	Function
LF Bode Box	Electronic injection transformer, DC-1 MHz	Couples and isolates AC source from circuit under test. Replaces Bode Box Models: 200-000 to -004
GP Bode Box	High frequency injection transformer	Couples and isolates AC source from circuit under test. Useful frequency range 1 kHz-10 MHz.
RLC Box	RLC Meter	Test fixture and software to automatically measure resistance, inductance, and capacitance of up to three components.
2249-C-60	BNC-BNC Cables	Connect FRA to injection transformer.
5187-C-60	BNC-Minigrabber Cables	Connect FRA or injection transformer directly to circuit under test.
3782-36-X	Banana-Minigrabber Cables	Connect electronic injection transformer to circuit under test.
IOZ-50 IOZ-100	Input/Output Impedance Transformer. Also used for conducted susceptibility testing.	High power injection transformer for injecting up to 200 watts of AC power into lines carrying DC current of up to 50 or 100 amps depending on model selected.

Calibration of Oscilloscope Probes for Flat Frequency Response

To read voltages higher than the hardware limits listed below or to minimize probe loading on a sensitive part of a circuit, you can use ordinary attenuating oscilloscope probes on the FRA inputs.

Model	Voltage Limit
51xx	100 Vpk

These probes have a series resistor (9 mega ohms in the case of a 10:1 probe) paralleled by a variable capacitor. The inputs of the FRA are 1 mega ohm, just like an oscilloscope, so the probes work on the FRA just as they do on an oscilloscope. Also, as on an oscilloscope, the probes have to be "calibrated". As with a scope, the term "calibration" means adjusting the variable capacitor that is in parallel with the series resistor so that the capacitive divider ratio is the same as the resistive divider ratio. The inputs of the FRA have a capacitance of about 50 pF, slightly more than most oscilloscopes, so the calibration for an oscilloscope will be slightly different than for an FRA. Here are the steps for calibrating a probe:
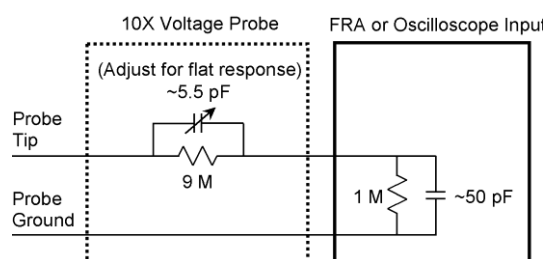
The objective is to treat the probe as a circuit you are trying to measure the frequency response of, then to adjust the variable capacitor so the attenuation at high frequency (determined by the ratio of the capacitors) matches the attenuation at low frequency (determined by the ratio of the resistors). The transition between low frequency and high frequency is typically in the range of 1-3 kHz. To do this, connect one of the BNC-Minigrabber cables provided to the FRA oscillator output. Connect the probe to the input channel you plan to use it on, and then connect another BNC-Minigrabber cable to another input channel. If you plan to use more than one probe, calibrate them one at a time on the input channel they are to be used on.
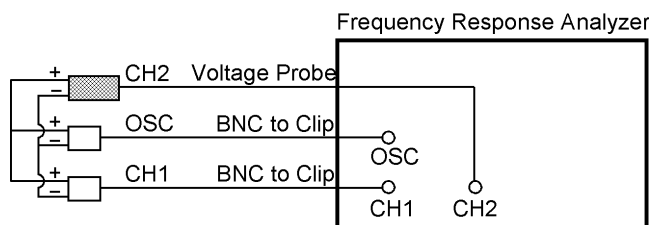
Connect all the grounds together and connect both the red Minigrabbers to the probe tip. For example purposes, we will assume that the probe is on channel 2 input and the Minigrabbers are on the oscillator output and channel 1 input.

Open the Venable System software. Set the sweep from 10 Hz to 100 kHz, log sweep at 10 steps /decade. The voltage is not critical.

You may want to set the analyzer to a 1.414 volt sine wave because the oscillator is specified in volts peak so the input channel will read 1 Vrms in the Readings window.

When the sweep parameters are set, click the "Take Data at Start Freq" button and let the analyzer take two or three readings. Check the Readings window to make sure everything is working. For example, if you used the same settings and channels as the example, the Frequency will be 10 Hz, Channel 1 will be 1 Vrms, and Channel 2 will be 0.1 Vrms if you are using a 10:1 probe. If everything is OK, click the "Run Sweep" button.

If the probe is not calibrated correctly, there will be a small step in the attenuation at the frequency where the divider changes from a resistive divider to a capacitive divider, usually in the range of 1-3 kHz. More noticeable than the gain step will be a phase "bump" or "dip" at the same frequency. The phase will have a "bump" if the high frequency gain is higher and a "dip" if the high frequency gain is lower.

When the sweep is complete, note the frequency of the peak or valley of the phase curve. You can add a slide bar to make it easier to measure the frequency.

Set the start frequency to the frequency of the peak or valley of the bump or dip in phase. Click the "Take Data at Start Freq" button and note the Phase reading in the Readings window. It should be the same as the peak or valley phase reading on the gain-phase graph.
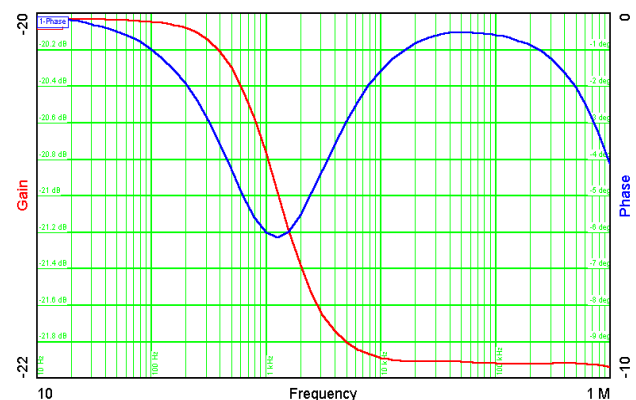
Uncalibrated probe
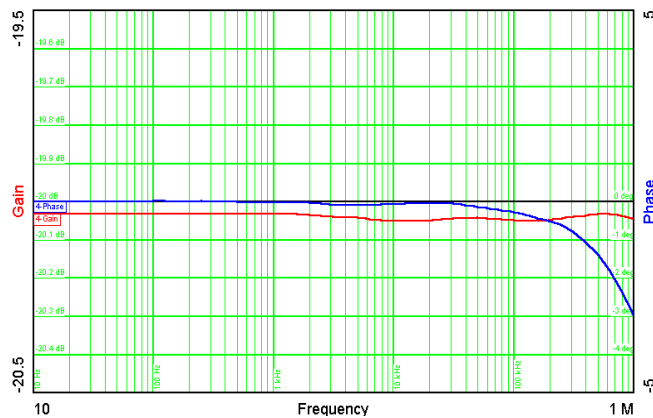
Turn the adjustment screw on the probe to set the phase as close to zero as possible. Zeroing the phase at the corner frequency automatically flattens the gain curve. If you want, you can do another sweep from 10 Hz to 100 kHz to check your handiwork.

Transfer function of calibrated probe. The gain and phase deviations at 1 kHz are minimized. Note the narrow gain and phase ranges of the graph.

Measurement of the Loops in a Multi-Loop Power Supply

By multi-loop, we mean there are several paths for the signal to take as it goes around the circuit, essentially creating a number of different loops which have some parts in common and other parts as separate parallel paths. This is common in power supplies that use a TL431 and optocoupler to cross the isolation barrier, in power supplies that use feedback from more than one output, and in mag-amp post-regulator circuits.

This is not to be confused with the term "multi-loop control" coined by Virginia Polytechnic Institute (VPI) to describe current-mode control. There are a number of viewpoints about the significance of the current loop, exemplified by the controversy between VPI and the California Institute of Technology (Cal Tech) on this subject. The view here at Venable Instruments is that the current loop simply forms a transconductance gain block in the overall loop and can be treated as one more gain block in series with a number of other gain blocks in the overall loop.

"Untestable" power supply has no single path meeting impedance criteria.
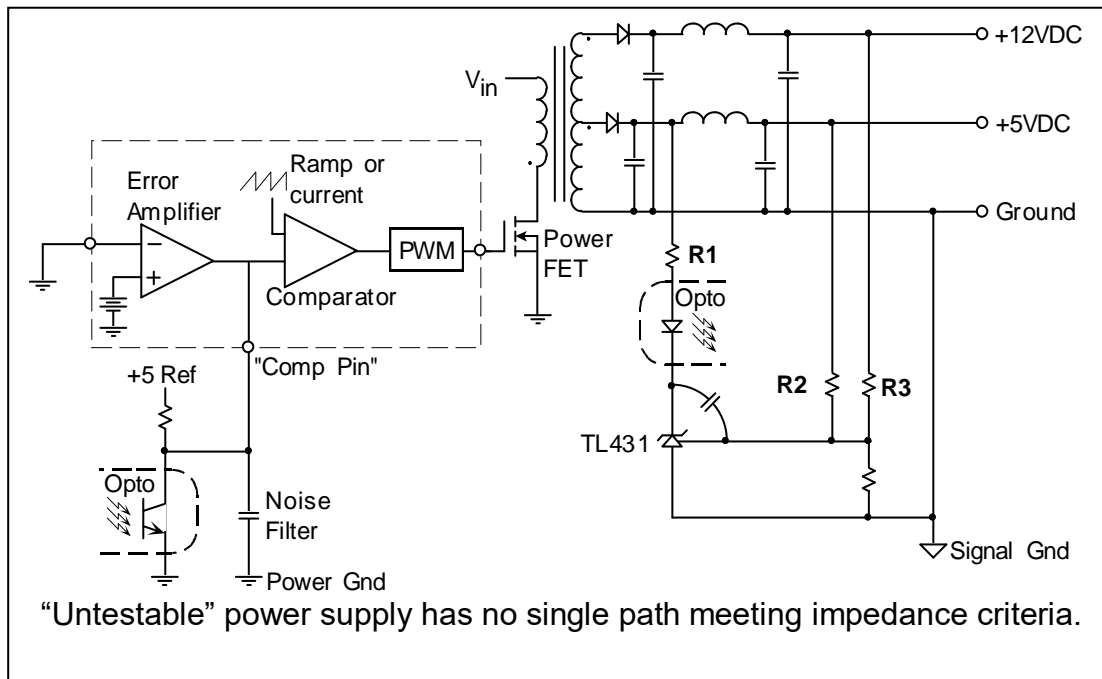
The above figure shows an "untestable" power supply. Unfortunately, this topology is not that unusual. There is no single path that meets the criteria of coming from a low impedance and driving a high impedance. The path into the "Comp Pin" of the PWM chip might qualify, but it is hard to be certain because the source impedance is high and the input impedance of the "Comp Pin" is unspecified. In addition, this point is usually very noise sensitive.

On the secondary side there are three paths that meet the impedance criteria, but all three of these paths are in parallel. There are two paths that most people recognize: the 5 V loop through R2 and the 12 V loop through R3. It is common to take part of the

feedback from the 5 volts and part from the 12 volts so that instead of having one output be perfectly regulated and the other vary 10% or so with line and load changes, both outputs vary by about 5%. The path that is not widely recognized is the path through R1. We call this the "fast loop" because it is the loop that dominates at high frequency. This path originates because the gate (control pin) of the TL431 is a virtual ground, and the capacitor connected from there to the cathode makes the cathode a virtual ground also at high frequency. Any AC voltage on the 5 V output modulates the current in the optocoupler diode directly through R1 by virtue of the fact that the branch ties between the 5 V output and a virtual ground. This direct modulation of the current in the optocoupler diode is much faster than the signals through R2 and R3, especially in the presence of the capacitor from cathode to gate.

In this topology example, there is a second L-C filter after the power stage to further reduce switching ripple at the output. This is also common, but the phase shift through the second L-C filter makes it difficult if not impossible to cross the loop over at a high bandwidth while sensing directly at the output of the power supply. Some designers solve this problem by connecting the "fast loop" through R1 to the junction between the power stage and the second L-C filter as shown in the example, avoiding the excess phase shift in the "fast loop". The "slow loop" through R2 and R3 provides good DC regulation. The AC loop response does not compensate for the second-stage L-C filter, but the fact that the loop bandwidth is much higher that would otherwise be possible means that the overall performance may actually be improved over a slower loop with the "fast" path tied to the output after the second-stage filter.

The problem with this topology is that there is no common point between the fast and slow loops as there is in the example given earlier in the basic part of this manual. If there was, at least the fast and slow 5 V loops could be combined into a single path. In this case, all three of the loops are separate. We solve this problem by measuring the loops separately and then combining them mathematically using the "Do Math on Data" window. One of the options in this window is called "Double to Single Loop". Parallel loops can be combined mathematically by the formula:

$$LOOPGAIN = \frac{GAIN\,1 + GAIN\,2 - 2 \times GAIN\,1 \times GAIN\,2}{1 - GAIN\,1 \times GAIN\,2}$$

This is the formula implemented in the "Double to Single Loop" function of the "Do Math on Data" window. When there are more than two loops, as in this case, the loops can be combined in pairs and the result combined with the additional loops one at a time.

When the true loop gain is high, the product of GAIN1 x GAIN2 will be close to 1 and the denominator of the equation will be small. The slow loop and fast loop will both cross the 0 dB gain axis at approximately the same frequency. With this formula there is the problem of a small difference between two similar numbers since at low frequency the term GAIN1 x GAIN2 is very close to +1. At low frequency the denominator of the equation is small since it is the difference between +1 and another number very close to +1. This causes some inaccuracy in the gain measurement when the gain is very high,

but near loop crossover the gain is small and the small difference of large numbers problem becomes insignificant. You can judge for yourself the effect from the examples below which are real test data from a production computer power supply.

The figure to the right is the "slow" loop measured at the top of R2. This is actual test data from a 5 V power supply using a TL431 and optocoupler in the feedback path. The loop crosses over about 60 Hz with over 90 degrees of phase margin.
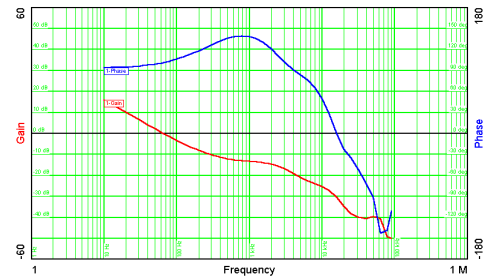
The figure to the right is the "fast" loop measured at the top of R1. This is also actual test data from the same 5 V power supply with the same TL431 and optocoupler in the feedback loop. There is no gain in this loop at low frequency because all the low frequency gain comes from the "slow" loop. This loop has a bandwidth of 6 kHz with about 90 degrees of phase margin.
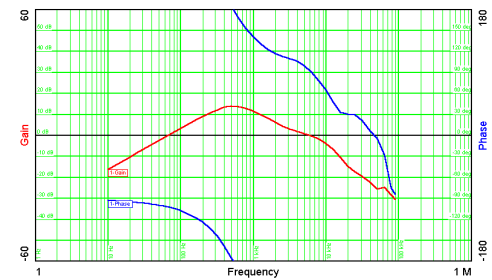
Measurement of the Pieces of a Loop

One of the most powerful techniques for analyzing and evaluating a circuit is to model it and then compare the model result with test results. Usually they will not match the first time. If a circuit is complex, like the one below, the loop is made up of many pieces. When the model and test results do not match, it is difficult to tell exactly which piece of the loop is causing the problem. The way around this dilemma is to test each piece of the loop separately, then compare test data from each piece of the circuit to the model results for that equivalent piece. If the pieces match, the model is probably accurate for that particular piece. If the pieces do not match, then you know which part of the model needs to be improved upon. The circuit is never wrong, it does what it does. If the model does not match, it is the model that needs fixing. It is possible that the circuit is not designed or built correctly and both the circuit and the model have to be changed, but an essential first step is to have the model results match the test data.

The loop in the example to the right is broken into 6 pieces. What you define as a piece of the loop is arbitrary, but it is helpful to break it up into the smallest functional blocks that can be conveniently measured and analyzed. One thing not shown on the above diagram is the signal injection point. A typical injection point would be in the trace where the filtered output voltage ties to R5 and R7, but you can inject anywhere. The FRA inputs are connected to the input and output of the gain block being tested.
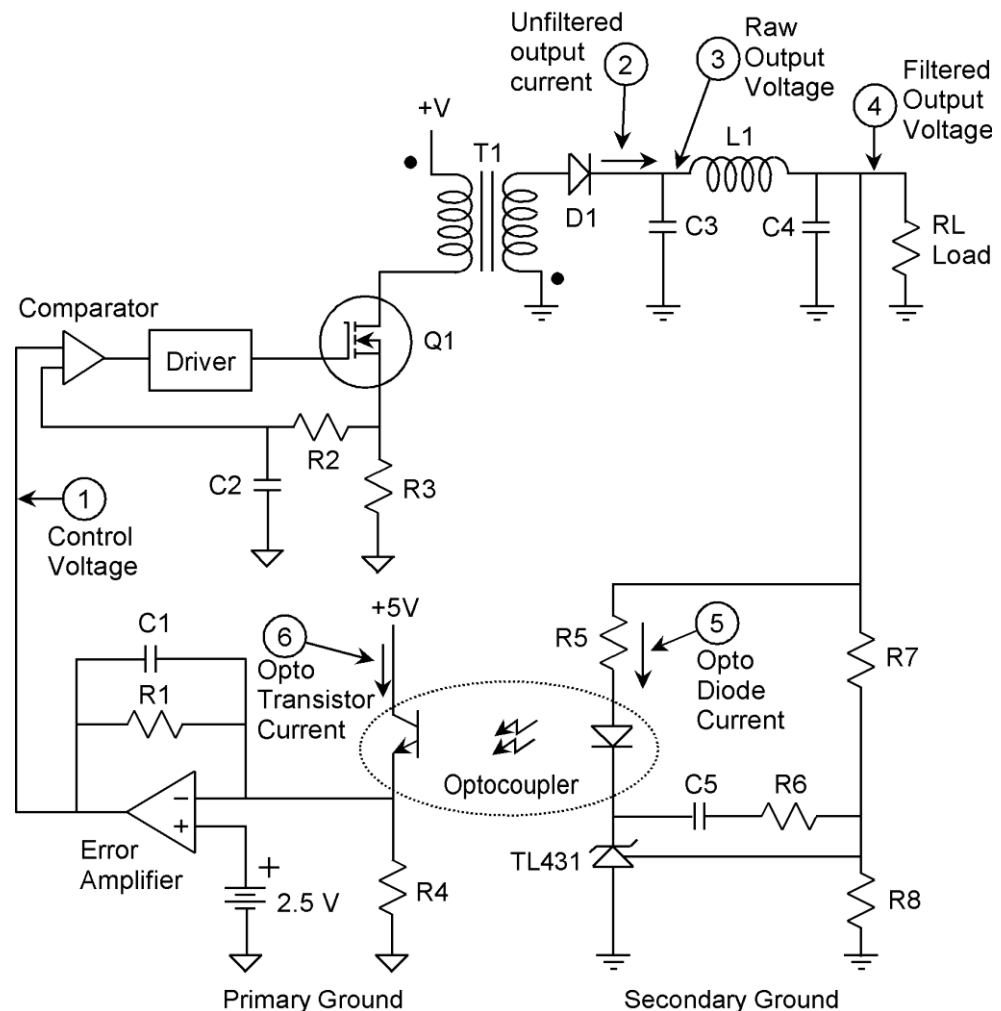
The six pieces of the loop are:

1) The transconductance of the power stage from control voltage in to unfiltered DC output current. The input is the compensation pin of the PWM integrated circuit and the output is the current in the rectifier diode measured with a current probe.

2) The unfiltered DC output current to raw output voltage. The input is the current in the rectifier diode measured with a current probe and the output is the voltage across the first set of filter capacitors.

3) The raw output voltage to filtered output voltage. The input is the voltage across the first set of filter capacitors and the output is the DC output voltage of the power supply.

4) The filtered output voltage to current in the opto-coupler diode. The input is the DC output voltage of the power supply and the output is the current in the opto-coupler diode. This current can be measured with a current probe or by analyzer specific techniques.

   For models with floating inputs, such as the 350a/350b/220/250, 3120, 350c, 63xx, 74xx and 88xx analyzers, you can measure the voltage directly across R5, taking advantage of the fact that the input channel returns of the Venable Models float.

   For models 3215/3225/3235 you can measure the voltage directly across R5 using channel 1 configured as a differential channel as long as the voltage doesn't exceed 30 V on either input.

   The input channel returns of the HP3577, 43xx and 51xx series analyzers are at ground so the voltage across R5 can't be directly measured with this analyzer unless you use differential amplifier or probe connected to the input.

5) Current in the optocoupler diode to current in the optocoupler transistor. This is called the current transfer ratio, CTR. It is constant at low frequency but rolls off at high frequency. The input is the current in the optocoupler diode, measured as described in block 4 above. The output is the current in the optocoupler transistor. The output measurement requires a current probe, and most likely cutting a circuit trace. It is not strictly necessary to measure this particular gain block, since all the AC current through the transistor has to flow through the parallel combination of R1 and C1. The inverting input of the error amplifier is a virtual ground and no AC current flows through R4. The reason you might want to make this particular measurement is to be able to separate the parasitics of the optocoupler from the parasitics of the error amplifier so you can make your model more accurate.

6) Current in the optocoupler transistor to control voltage out of the error amplifier. The final block has the current in the optocoupler transistor as the input and the control

voltage (output of the error amplifier) as the output. This gain should be just the optocoupler transistor current times the parallel impedance of R1 and C1, but the gain-bandwidth product of the error amplifier is often an important parasitic affecting the gain of this particular block.

Some of these pieces require the use of a current probe. The output of a current probe is not 1 volt per amp unless the current probe is set to the particular range that gives this scale factor. You can use the "Scale Factor" provision of Data Set Properties to scale the data for the particular current probe gain setting so that the output is always 1 amp per volt.

Measurement of the Impedance of Components versus Frequency

There are two ways to measure impedance of components. We offer an RLC measurement package that has a test fixture and a special menu to correct for measurement fixture parasitics after the component measurement is complete. This is the first and easiest way to measure the impedance of components.

The second way, the method described here, does not require any additional hardware or software except a single resistor. Although, you can use our RLC text fixture if you have one. It has the same connections as shown in the figure below. The ability to measure and display impedance is a built-in feature of the Venable software package. This includes the ability to measure parasitics of test fixtures. The Do Math on Data window can be used to correct for test fixture parasitics. This procedure is not difficult but it is not automatic the way it is in the RLC measurement menu.

Impedance is voltage divided by current. We use the oscillator output of the FRA as a signal source. We connect the oscillator to the component under test, usually through a series resistor. You can measure the current with a current probe, but it is usually easier to take advantage of the floating inputs of the analyzers (except the 43xx) and measure current by measuring the voltage across the resistor in series with the component under test. For the 43xx, you can use a differential amp or an analyzer with floating inputs to measure current by measuring the voltage across the resistor in series with the component under test. There is about 200 pF of stray capacitance from return to chassis and about 50 pF of capacitance from input to return of the analyzer inputs. For this reason, we usually connect the current measuring channel (CH1) "backward", so the return is connected to the oscillator output as shown in the figure. This minimizes the parasitic capacitance across the component under test.
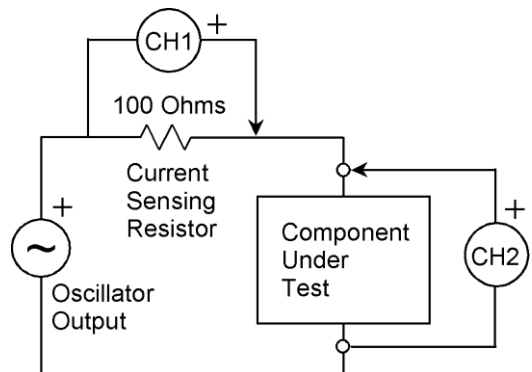
Let's measure the impedance versus frequency of an electrolytic capacitor as an example. The first thing to do is find a good resistor to sense the current. It should be non-inductive, such as a film resistor. The value is not critical but a value around 100

ohms will give a good balance between the impedance being tested and the open circuit and short circuit parasitic impedances. Although the particular value is not critical, it *is* important to know the absolute value of the current measuring resistor. You should measure the value with a good ohmmeter before you start or use a resistor with a tight tolerance.

Connect the oscillator output, channel 1 input, and channel 2 input as shown in the figure above or connect the RLC test fixture to the analyzer. Open the Venable System software and adjust the settings to match the sample screen on the previous page the Analyzer Control window. . We used the maximum output voltage (10 V peak for Venable models) to get the best accuracy and Medium integration time for the best speed (2.5 sec for 63xx). Before starting the sweep, make sure the graph screen displayed is a blank impedance plot. You can open a new one when you click the File-New and then click on Impedance and Phase plot. Then, after adjusting the Analyzer Control settings, click the "Take Data at Start Freq." button. When the analyzer has taken a few data points, click the "Run Sweep" button.
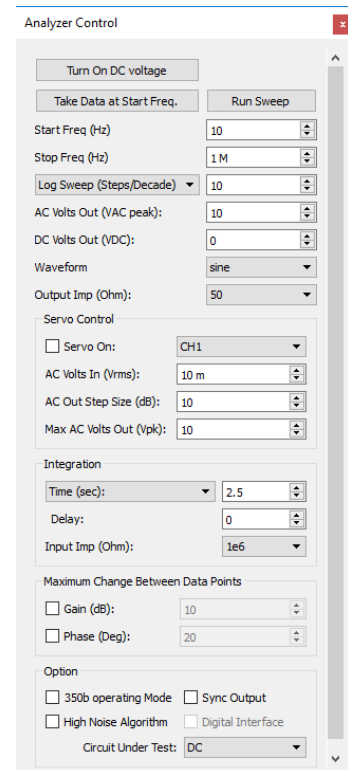
At the bottom of the plot is a sub-window displaying the Channel Ratio and Scale Factor. You can only change this setting if data has already been plotted on the graph or after the sweep has started. If you made the same connections as the test setup schematic on the previous page or are using the RLC test fixture, the Channel Ratio will be CH2/CH1 (the default value), but the Scale Factor has to be changed to match the value of the current sensing resistor. In this case, with the current measuring channel connected "backward", the polarity of CH1 will be "backward" also. You correct this by entering the current sense resistor value as negative. If you used 100 ohms to measure the current, enter the Scale Factor as –100 in the Data Set Properties sub-window as shown above. If you are using a current probe, the Scale Factor is the Volts/Division divided by the Amps/Division of the current probe amplifier.
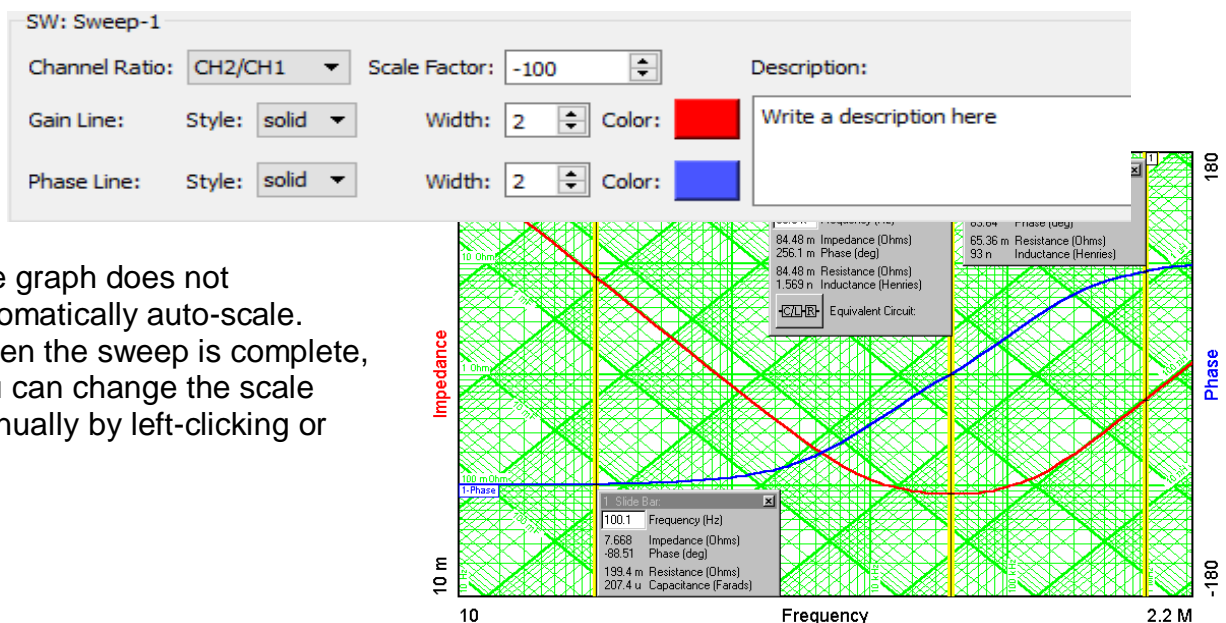
The graph does not automatically auto-scale. When the sweep is complete, you can change the scale manually by left-clicking or

right clicking and dragging the mouse on any axis of the graph to get the desired minimum and maximum values. In the example shown, we were testing an aluminum electrolytic capacitor with a rated voltage of 35 VDC and a nominal capacitance value of 200 µF. We added slide bars to show the capacitance at 100 Hz (207.4 µF), the ESR when it is purely resistive (84.48 milliohms), and the inductance at 1 MHz (93 nH).

The only problem with the graph on the preceding page is that the parasitics of the test setup are not taken into account and may be significant at low or high impedance levels. To check on this, let's measure the parasitics. To do this, simply make the same measurement as before, except do it with the Component Under Test being first an open circuit and then a short circuit. Here are the results of those two measurements.

For clarity, the plot type below is impedance only.
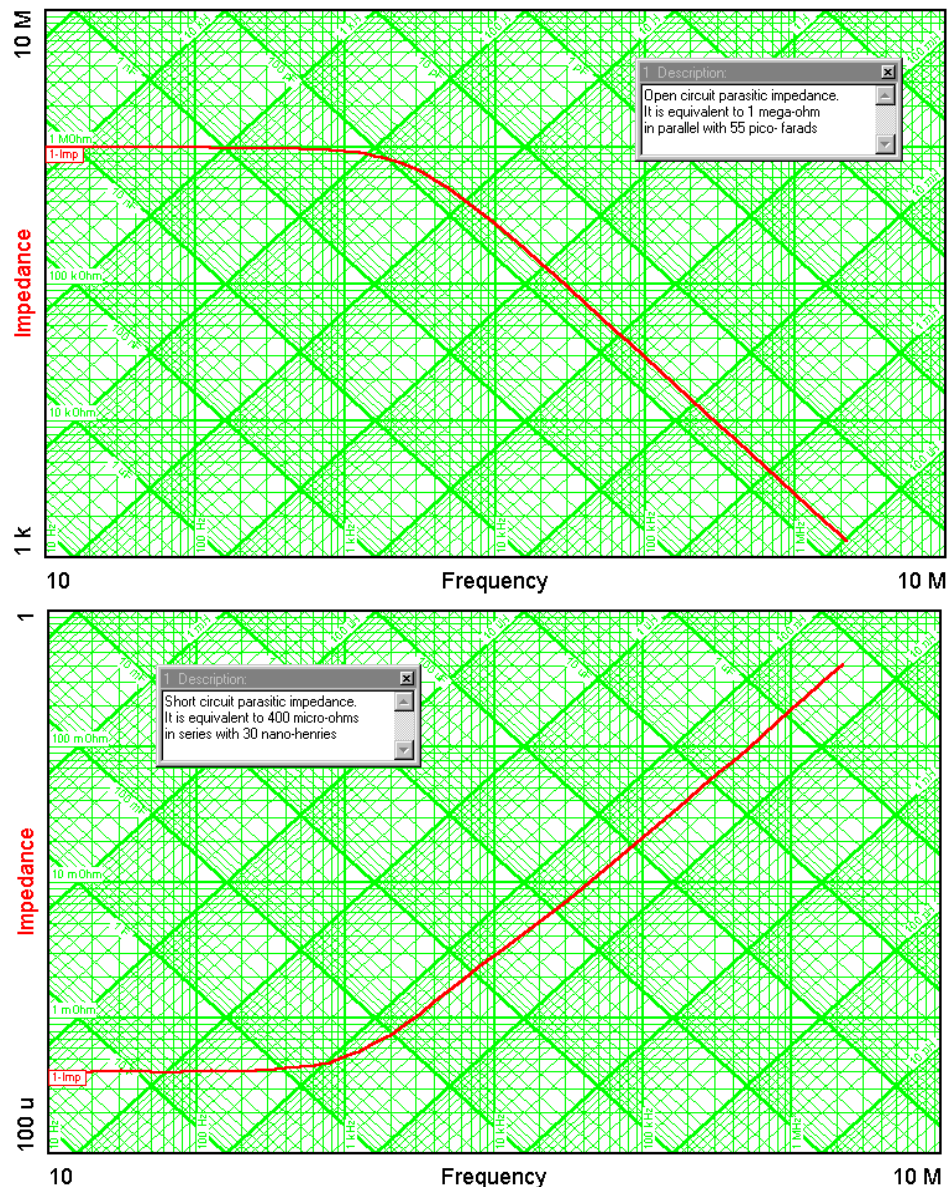
The graph below shows the result of using "Math Options for Impedance Data" to subtract the parasitic short circuit impedance from the capacitor impedance test data. The impedance is too far below the open circuit impedance to require compensation for open circuit parasitics. The true inductance of the capacitor can be seen in the text box near the slide bar to be 59.21 nH instead of the 93 nH we measured without correcting for short circuit parasitic impedance.

Math Options for Impedance Data

Left Operand
- ● Data Set 1-SW: Sw ▼
- ○ Scalar 1

Right Operand
- ● Data Set 2-SW: Sw ▼
- ○ Scalar 1

Action
Subtract ▼

Plot

3 Description:
This graphs shows the capacitor impedance test data superimposed on the short circuit impedance data. "Do Math on Data" was then used to subtract the parasitic impedance from the test data. The result is highlighted.

3 Slide Bar:
1.003 M Frequency (Hz)
376.7 m Impedance (Ohms)
82.15   Phase (deg)
51.43 m Resistance (Ohms)
59.21 n Inductance (Henries)
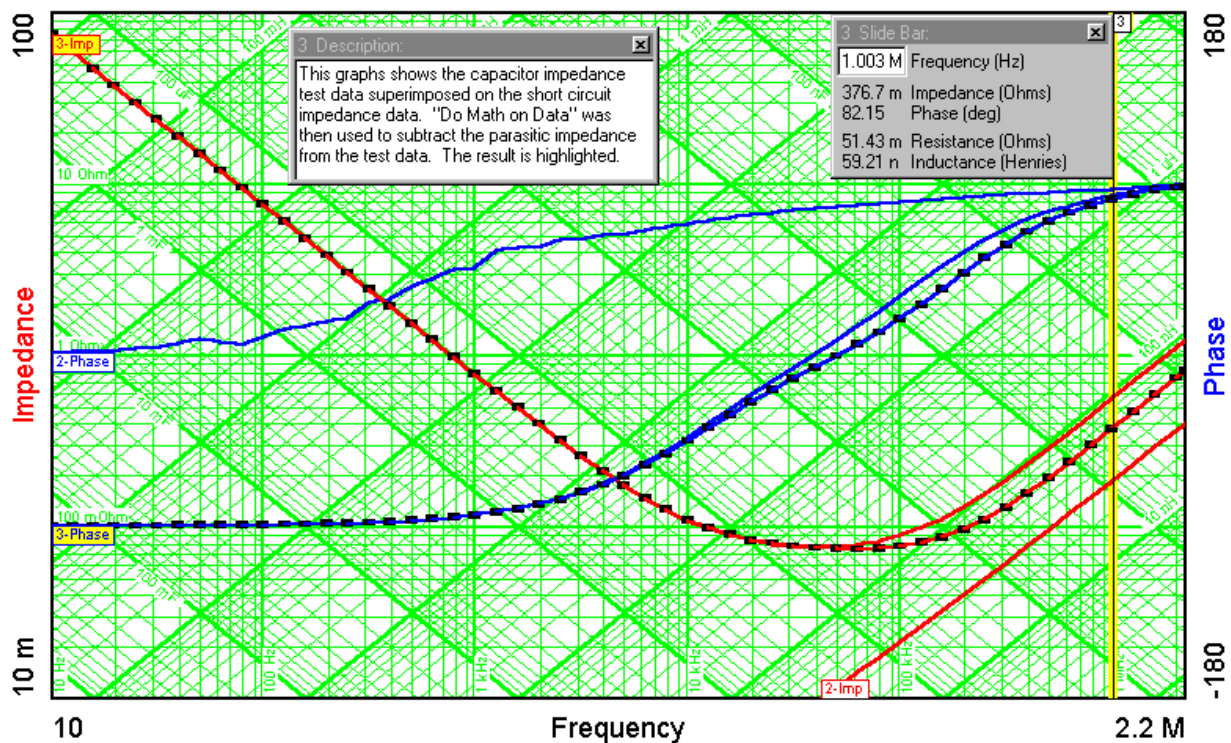
The example on the previous pages was for measuring a low impedance and correcting for short circuit parasitics by subtracting the short circuit impedance from the test data. The same concept can be used for measuring high impedances and correcting for open circuit parasitics, except it is a little trickier because the open circuit parasitics are in parallel with the test data instead of in series.

The best way to illustrate the difference is with an example. Let's say we want to measure a 10 pF capacitor. This is a lot smaller than the parasitic capacitance of approximately 60 pF for an Analyzer and the impedance of a 10 pF capacitor is greater than the parasitic resistance of 1 mega ohm for all frequencies below 16 kHz. To do the test, first connect the 10 pF capacitor as the "Component Under Test" in the test setup on page 36 and run a sweep as before choosing "Impedance and Phase" as the plot type. Use the Analyzer Control settings to the right and the resulting plot with the scales changed to accommodate the data is shown below. Choose "Impedance and Phase" as the graph type and choose Channel Ratio = CH2/CH1, and Scale Factor = -100 if you are using a 100 ohm resistor to measure the current and the connection is the same as the figure on page 47 or if you are using the RLC test fixture.

The graph to the right shows the impedance versus frequency plot of the 10 pF capacitor together with the parallel parasitic resistive and capacitive admittance. So far, it just looks like 1 mega ohm in parallel with about 70 pF, but we would like to look at the 10 pF capacitor

by itself, not in parallel with all those parasitic elements. To do that, we need to run another sweep and then use "Math Options for Impedance Data" to fix the plot.
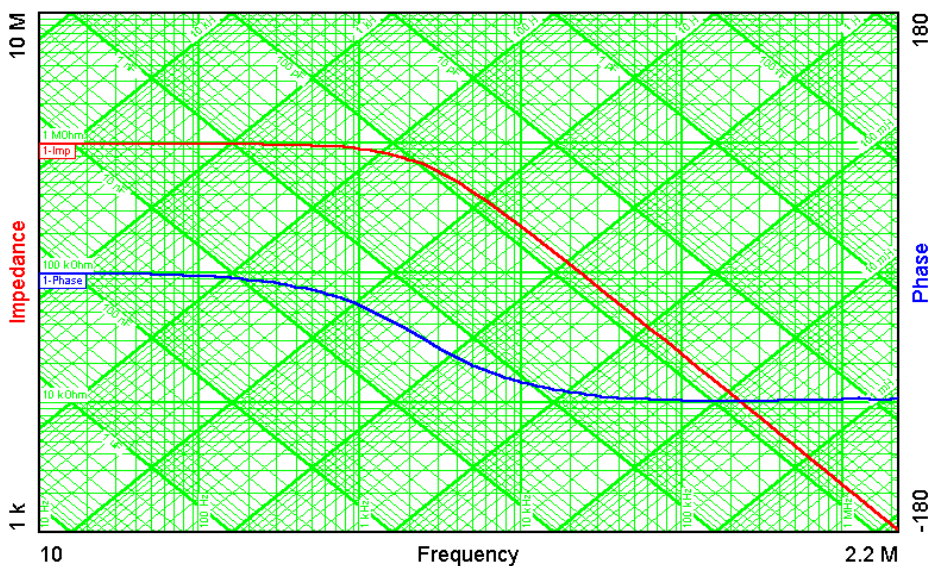
Disconnect the 10 pF capacitor from the test setup but leave the 100 ohm resistor and leave the oscillator and both voltmeters connected as before. Run a second sweep with nothing connected except the voltmeters. This is the open circuit impedance plot. There will not be much difference between the two. The graph to the right shows both sets of data with the open circuit parasitic data highlighted (selected).
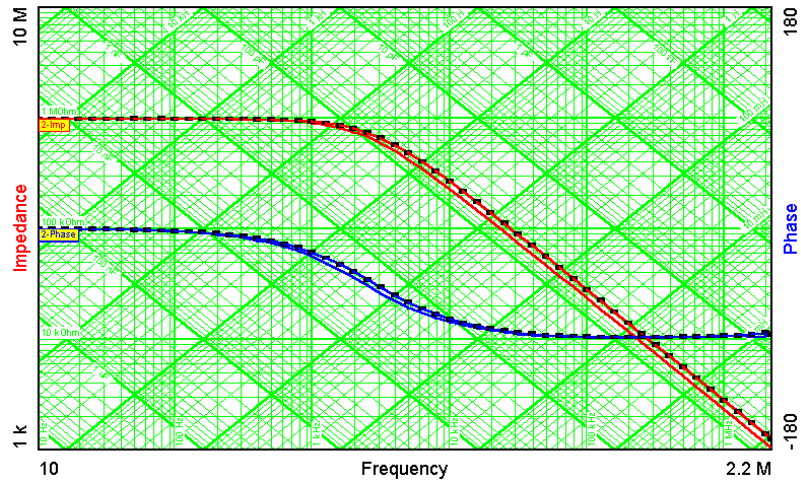
Now comes the tricky part. What you want to do is subtract the *admittance* of the open circuit parasitics from the *admittance* of the test data (which includes the impedance of the component under test in parallel with the parasitic open circuit impedance). We have a feature in "Math Options for Impedance Data" called "Subtract in Parallel". This will put the negative of the parasitic impedance in parallel with the test data, effectively canceling out the open circuit parasitic impedance and yielding only the impedance of the component under test.

The plot on the upper right shows the results of the step on the previous page. The highlighted curve is the result of doing a "Subtract in Parallel" of the test data with from the parasitic data.

The graph below is the final impedance versus frequency graph of the 10 pF capacitor.
The impedance data is accurate up to about 100 mega ohms. Beyond that, the data is noisy because the parasitic resistance is only 1 mega ohm. At the upper left corner of the graph we are trying to resolve the difference between 1 mega ohm and 1 giga ohm in parallel with 1 mega ohm, and understandably the result has a little variance because we are approaching the limits of measurement resolution.
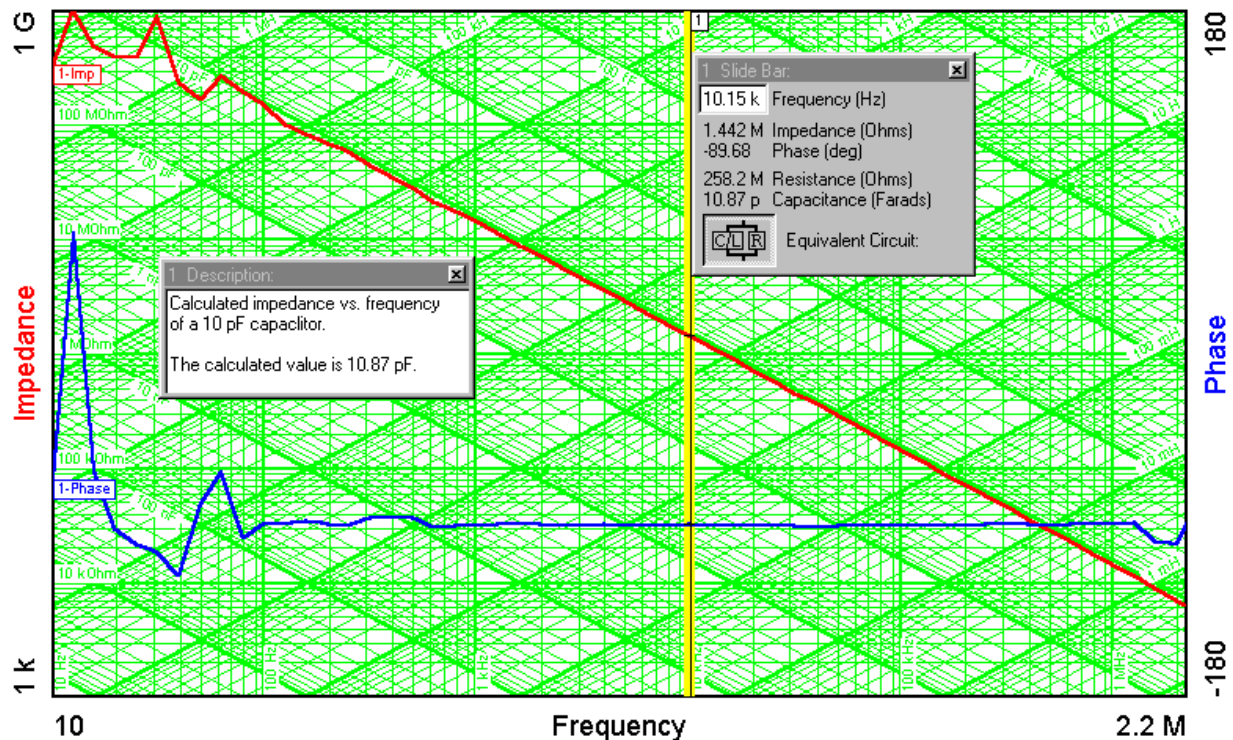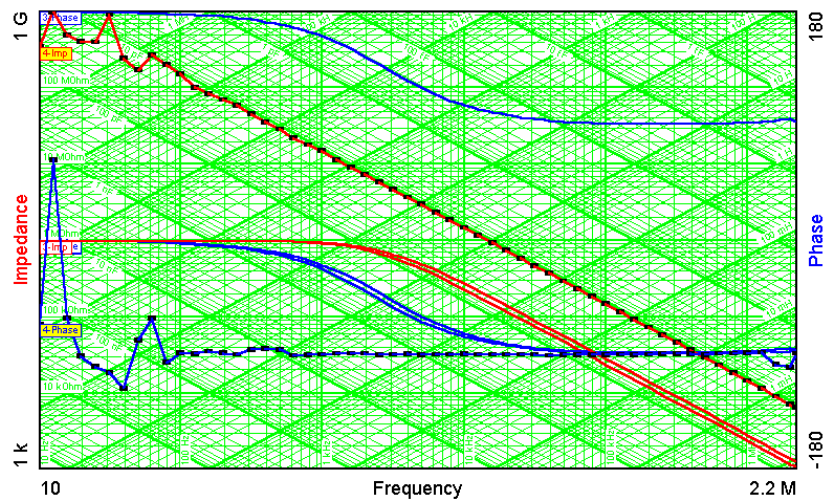
Measuring Impedance with the RLC Control Menu Option

Overview

The Venable RLC menu in combination with a supported Frequency Response Analyzer (FRA), allows you to measure the frequency response of passive components. You can calibrate your RLC fixture, run sweeps through a range of frequencies, build a theoretical model based on the empirical data and generate reports. This menu is enabled with the purchase of an additional RLC license for the Venable Stability Analysis software Ver. 6.1.

The RLC software is controlled mainly through the RLC Control Panel, which has three components: Measurement, Model, and Report.

Measurement Tab
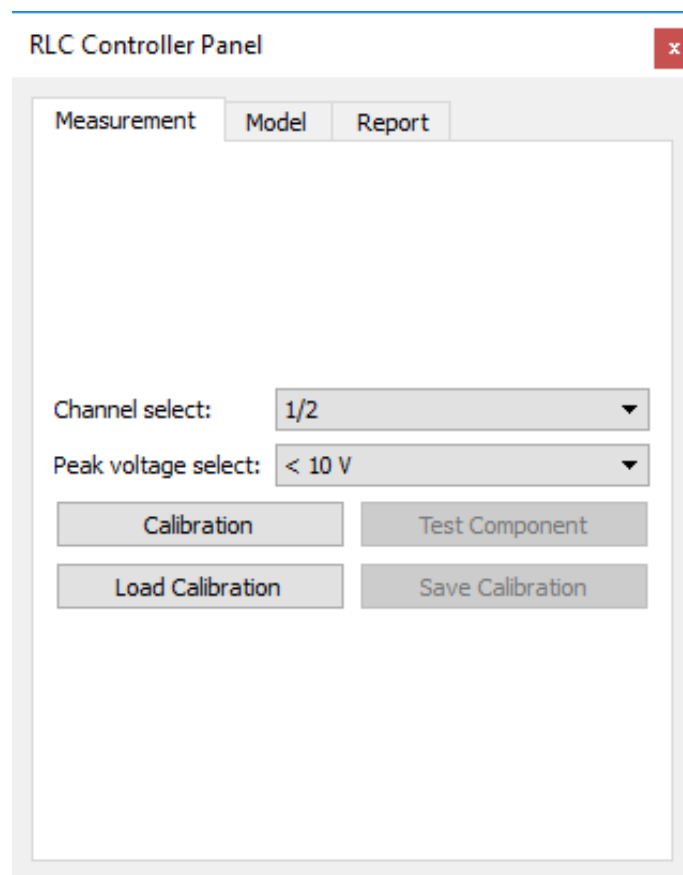
The Measurement tab

Before measurements can be taken, the system must be calibrated. The purpose of the calibration process is to quantify the parasitic components that are present in the RLC

fixture. With an estimation of the values of these parasitic elements, the actual measurements of RLC circuits can be corrected to improve measurement accuracy. The parasitic components are usually modeled as two elements: a low impedance in series (of inductive nature), and a high impedance in parallel (of capacitive nature).

Channel Select: *(Models 74xx and 350c only)*

Channel Select allows the RLC box to be used on alternative channel combinations for analyzers with 3 or 4 channels.

Peak Voltage Select: *(Models 43xx, 51xx, 63xx, 74xx, 88xx and 350c only)*

This feature was added to correct for channel input attenuation changes during auto-ranging in data plots at voltage levels above 10V. Peak Voltage Select should be set to the peak combined AC and DC voltage level to be measured. Slight variances, due to channel input attenuation settings above 10V on the input, requires this setting to be set correctly for the calibration to be done based on the combined peak voltage level. Once the calibration has been completed, this value cannot be changed without closing and reopening the application. On an analyzer with 3 or more channels, the setting can be changed when the channel select is changed to a new setting

Calibration
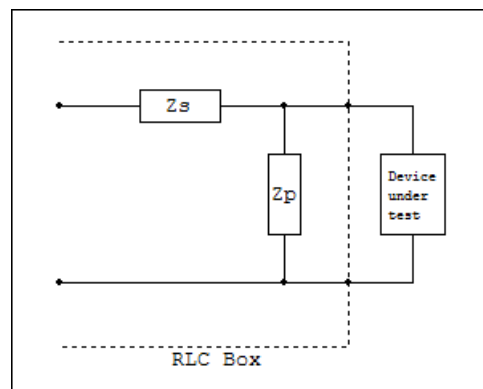
Parasitic model

The calibration process consists of two steps: an open-circuit sweep, and a closed-circuit sweep. Once the calibration process starts, the RLC software will prompt you to first open and then close the circuit in order to take the respective measurements.

Open Circuit Sweep

When the circuit is open, the impedance of the parallel parasitic element can be measured (the impedance of the series element is so much smaller that its effect is negligible in the measurement).
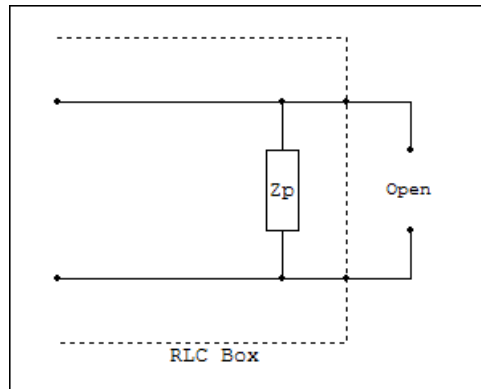
Open circuit model

Closed Circuit Sweep

After the open-circuit sweep is complete, a closed-circuit sweep is run. In this case, the parallel element can be removed from the equation by shorting it, and the impedance of the series element can be calculated.
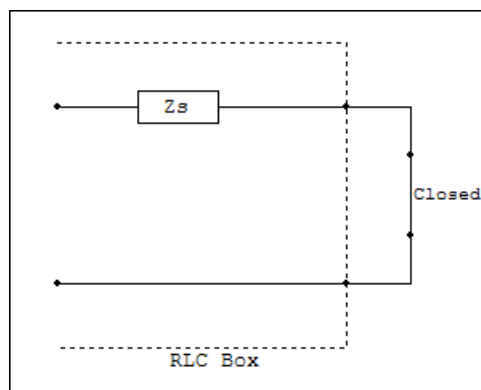
Closed circuit model

Load and Save Calibration

After the open-circuit and closed-circuit sweeps are complete, you can save and recall the calibration data for future use. That way, it is not necessary to re-calibrate when opening a new tab or a new file for a different set of measurements.
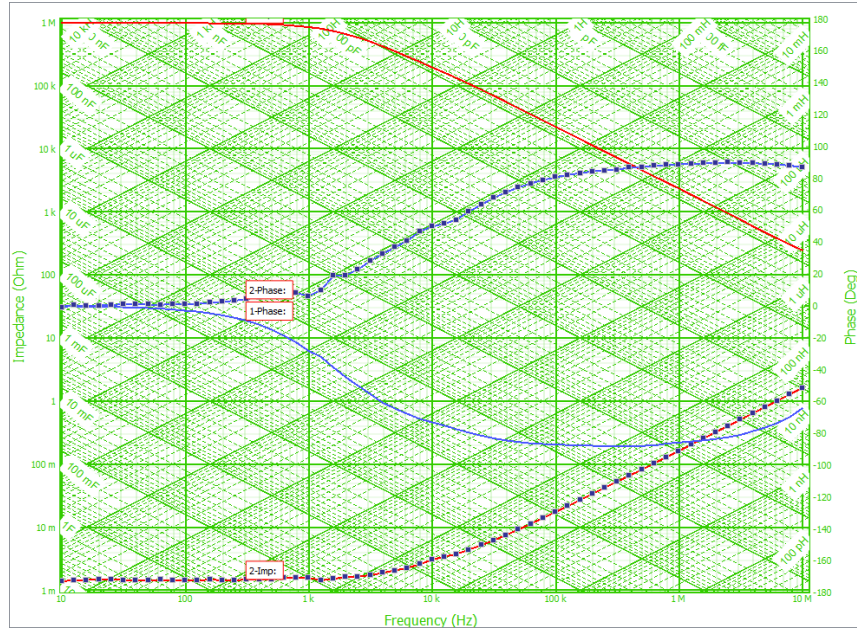
Example of open and closed circuit impedance and phase plots

Test Component

Once the calibration is done, the system is ready to measure RLC circuits. Use the "Test Component" button on the RLC menu to run the RLC sweep. The software will automatically correct the raw data and create another plot with the tag "w/o parasitics". This plot is the result of subtracting the series and parallel parasitic elements from the raw measurement and creating a corrected measurement plot with the parasitic elements of the RLC fixture removed.
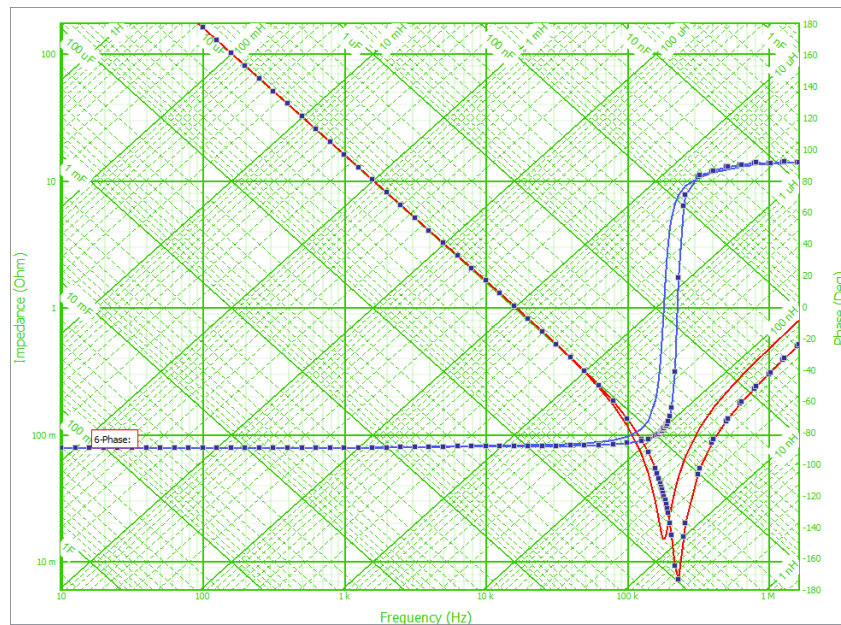
Uncorrected and corrected plots

The controls related to making measurements are set in the Analyzer Control Menu:

- Run Sweep: This control will run a standard sweep. No parasitic correction will be applied to the data. Use the "Test Component" button on the RLC menu to get a corrected plot of the measured component.

- Start and stop frequencies: they define the first and last points in the sweep. In the case of a single point setup, start and stop frequency are the same.

- Resolution: Sets the number of frequency points by steps per decade or steps per Hz.

- A/C and D/C voltages: controls that specify the AC injection voltage and bias.

- Integration time: specifies the length of time over which each sample is taken. A longer time setting will take longer to run but will produce more precise results.

- Limit maximum phase change between data points: without this option, the software will always take 10 data points per frequency decade, regardless of the results. A problem arises when a sweep involves a resonant point, where the impedance and phase change abruptly from one frequency to the next. The resonant point can be missed. In this case, you can use this feature to mitigate the problem. If, for example, you set the maximum phase change at 15°, the software will adapt its frequency interval so that no two points will be more than 15° apart in phase.

Analyzer Control		
Turn On DC voltage		
Take Data at Start Freq.		Run Sweep
Start Freq (Hz)	10	
Stop Freq (Hz)	10 M	
Log Sweep (Steps/Decade) ▼	10	
AC Volts Out (VAC peak):	3.16228	
DC Volts Out (VDC):	0	
Waveform	sine ▼	
Output Imp (Ohm):	50 ▼	

Servo Control
☐ Servo On:	CH1 ▼
AC Volts In (Vrms):	10 m
AC Out Step Size (dB):	10
Max AC Volts Out (Vpk):	10

Integration
Time (sec): ▼	2.5
Delay:	0
Input Imp (Ohm):	1e6 ▼

Maximum Change Between Data Points
☐ Gain (dB):	10
☑ Phase (Deg):	15

Modeling Tab

The modeling tab is used to create a theoretical model out of the circuit that was measured. You can choose the component type and the data set to model. The available component types are different combinations of a resistor, a capacitor, and an inductor, arranged in parallel or in series. After selecting the component type and data set, the calculated theoretical values for R, L, and C are displayed on the window. You can also override the values of R, L, and C by typing different values in the respective controls. You can create a plot that will represent the theoretical RLC circuit based on the values shown. If you wish to create a model with more than three components, you could create multiple models and combine them in series or in parallel using the math functions.

The theoretical values of the circuit components are calculated by attempting to fit the model to the measured data. In the case of components in series, the maximum or minimum values of susceptance (for capacitive or inductive circuits, respectively) are used to calculate the capacitance or inductance. In the case of components in parallel, the maximum or minimum values of reactance (for inductive or capacitive circuits, respectively) are used. In order to more accurately calculate the theoretical model, it is better to sweep through a wide frequency range.
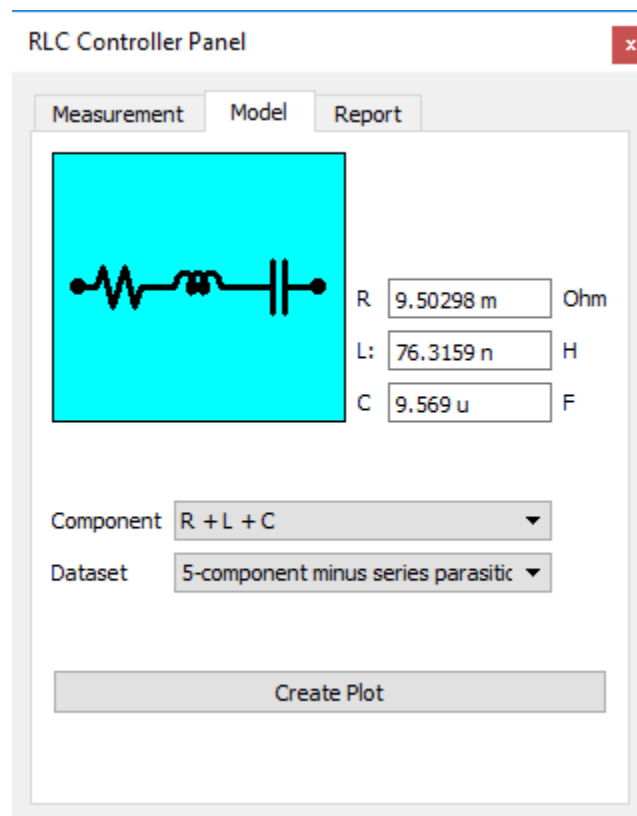
The Modeling Tab
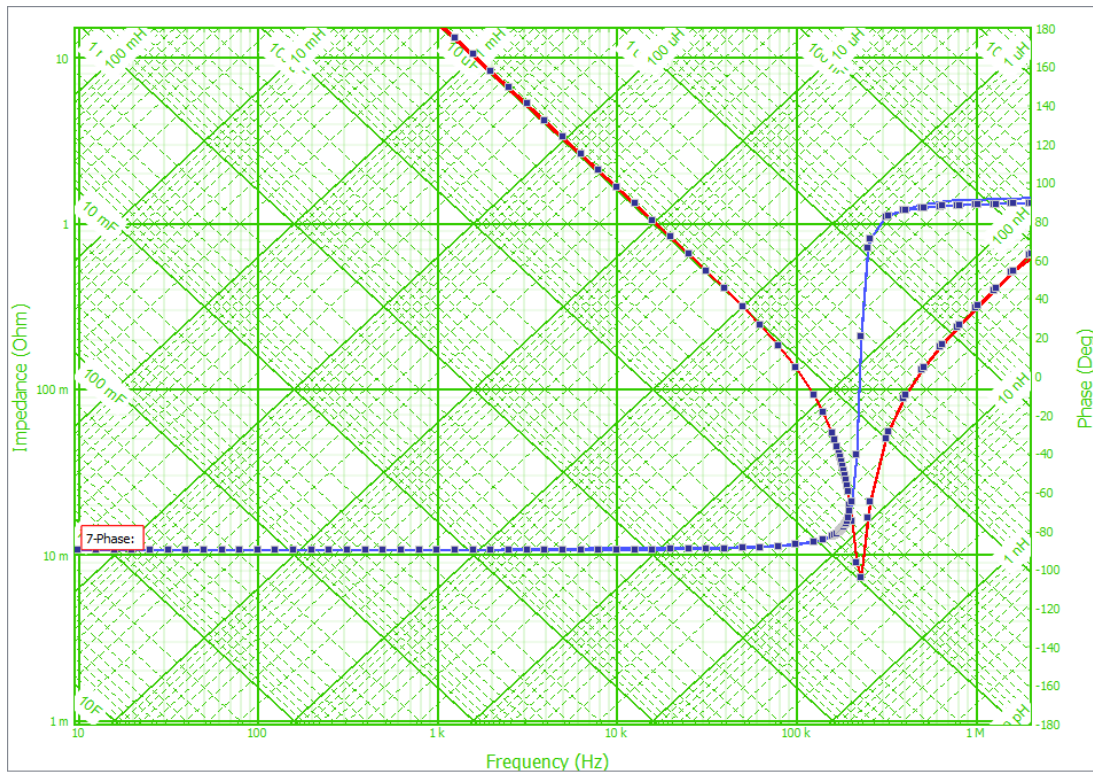
Example of a measured Capacitor and its derived model

Reporting Tab

The RLC application can generate reports in Microsoft Excel XML format, which is supported by Microsoft Excel version 2003 and later. The reports include the data in tabular format (optionally with voltage data), and information about the model. To enhance a report, you can copy a chart to the clipboard (with the "Copy Graph" menu item) and then paste it in an Excel spreadsheet. Also, the drawing of the circuit model can be copied by pressing CTRL-C while in the "Model" tab.

Alternatively to using the "XML report" feature, you can copy a dataset (first select it on the graph, then copy it with CTRL-C) from the Data Set Text Display tab and then paste the data in any text editor, word processor, or spreadsheet software.

NOTE: when you copy a dataset to the clipboard, you can control whether the data is copied as impedance/phase or as opposed to channel voltage/phase by using the radio button "Ratio" in the "Data Set Text Display tab.
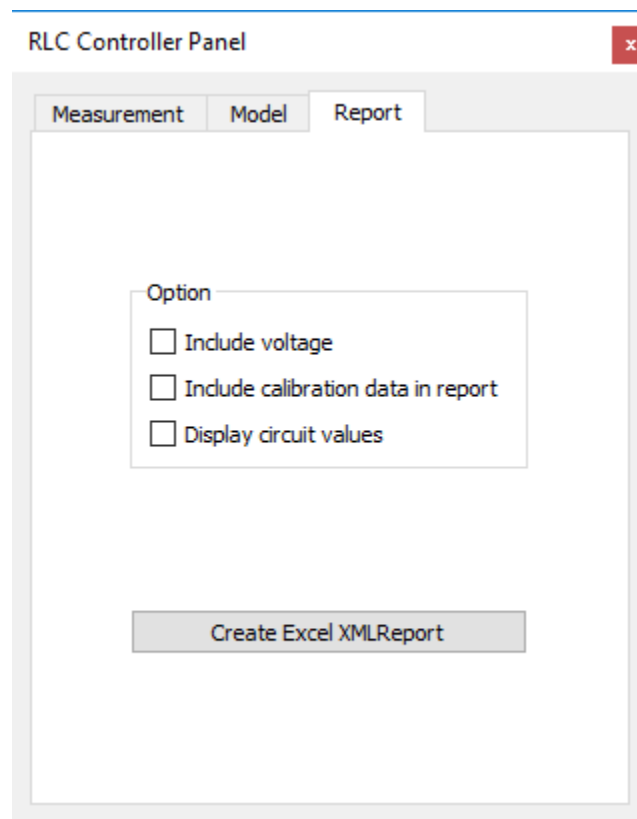
Reporting tab

Measurement of the Input and Output Impedance of Power Supplies

Why are Source-Load impedance measurements important? Distributed power systems can oscillate, especially when driving one power supply with another power supply. Loads, especially switching power supplies, draw constant power over the input voltage range and therefore look like negative resistors at low frequencies. Connecting a negative resistor to a resonant circuit can reduce or eliminate the damping. Negative input impedance can cause an entire system to become unstable and oscillate.
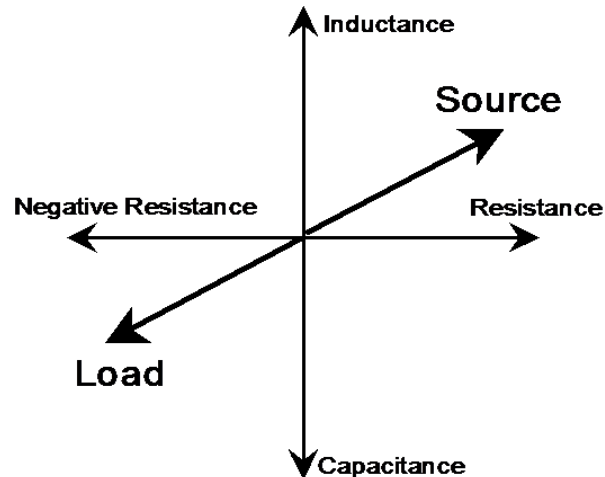
- Source Z + Load Z = Zero
- Magnitude of source impedance = Magnitude of load impedance
- Phase of source impedance = Phase of load impedance +180°

Criteria for Instability
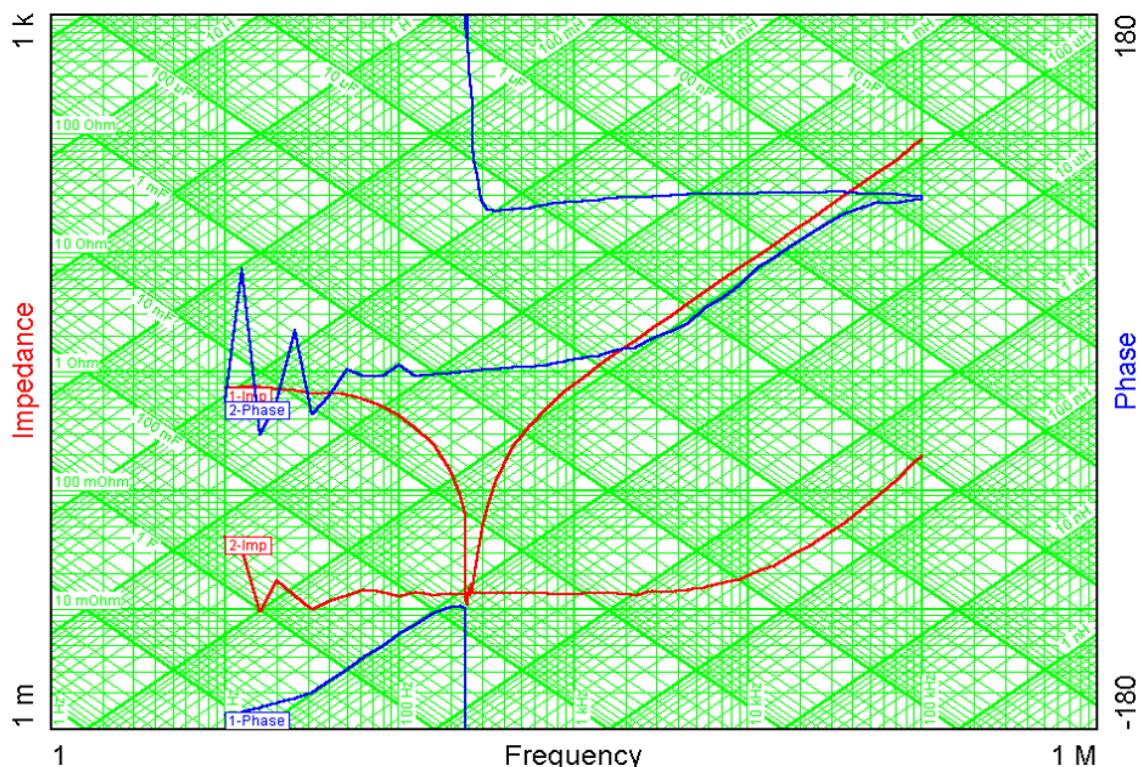
Unstable Source and Load Combination Over Plotted

04/14/20

72

With the Input/Output Impedance Test Set option of the Venable Instruments System, you can measure input and output impedance of any power supply (or almost anything else for that matter). The Input/Output Impedance Test Set consists of a high-power 4-quadrant amplifier and power injection transformer. The injection transformer is capable of handling large amounts of DC current while still coupling an AC signal into the circuit. The standard power amplifier is rated 300 watts, and higher power ratings are available on special request. The injection transformer is rated 50 or 100 amps DC depending on model. Because the power amplifier is 4-quadrant, this system will also work for making measurements with an AC power source instead of DC. Since AC load current is transformed by the injection transformer and has to flow through the power amplifier, check with us here at Venable Instruments before making high power AC measurements to make sure the power amplifier is not overloaded and damaged. DC measurements are safe up to the rated current of the injection transformer.
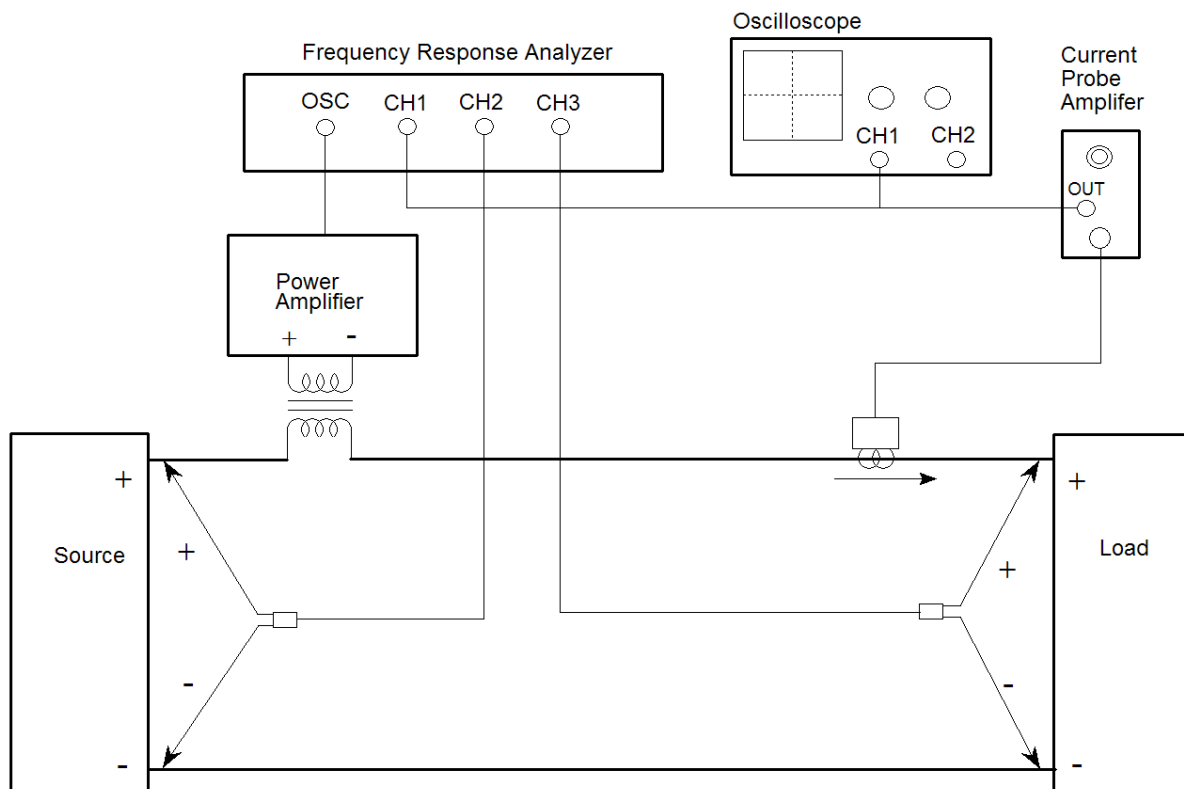
Test setup for measuring input and output impedance.

- CH3 = Voltage across the load
- CH2 = Voltage across the source
- CH1 = Voltage across R_{sense} or Voltage output of Current Probe Amp
- Source output Z   =   -CH2 x R_{sense} / CH1
- Load input Z   =   CH3 x R_{sense} / CH1
- Scale factor = R_{sense} or Effective Resistance of the Current Probe

The test setup above shows a general purpose setup for measuring impedance. If the "Source" is truly a source and the "Load" is the Unit Under Test, this setup will measure input impedance of the Unit Under Test. As with component measurements, the impedance is voltage divided by current. In the test setup shown, source output impedance is CH2/CH1 and load input impedance is CH3/CH1 noting that there may also have to be a scale factor if the output of the current probe amplifier is not 1 volt per amp. If the "Load" is the Unit Under Test, CH3/CH1 is the input impedance of the Unit Under Test.

If the "Source" is the Unit Under Test and the "Load" is an actual or simulated load, this test setup will measure the output impedance of the Unit Under Test. CH2/CH1 is the output impedance of the Unit Under Test. If your system does not have a channel 3, then you must move CH2 to the source side of the injection transformer to measure the output impedance of the Unit Under Test.

Current probes are directional. If the arrow on the current probe is pointing toward the load, then the measured impedance of the load will be the correct polarity. The scale factor of the data set will be the effective resistance of the current probe, which is the rated volts per division of the current probe amplifier output (usually 0.01 volts per division) divided by the amps per division setting of the current probe amplifier. For measuring the output impedance of the source, if the arrow on the current probe is pointing toward the load (away from the source) then the scale factor will be the negative of the value calculated above for the load input impedance data set. It is important to pay attention to the direction of the current probe arrow, especially if you do not have a feel for the expected phase of the test result readings.

Impedance measurements are a lot trickier than they look. The hardest part is getting the current probe set up properly. The second hardest part is making sure the drive levels are correct and nothing is being under driven or overdriven. The third hardest part is getting the scale factor and polarity correct.

Let's start with the current probe. If you want good performance at low frequency, choose a DC current probe such at the Tektronix AM503S System. The current probe is not part of the Venable System or the Input/Output Impedance Test Set. It must be purchased separately if you do not already have one. Models are available in 20, 100, and 500 amp versions. Try to use one matched as closely as possible to the current you plan to measure. Trying to measure a few milliamps with a 100 or 500 amp current probe will not give very satisfactory results. The next thing to remember is that the Tektronix current probe amplifier is made to work into a 50 ohm load. That means you need to put a coaxial 50 ohm termination on each input of the FRA that is measuring current. The termination goes on the frequency response analyzer end of the BNC cable, not the current probe amplifier end. Make sure the termination is really 50 ohms by checking it with a good ohmmeter. You would be amazed how many 50 ohm terminations are "burned up" from connecting them to high-power sources. The accuracy of the impedance measurement is dependent on the accuracy of the 50 ohm termination. <u>Failure to properly terminate the current probe is a common source of error</u>

<u>in impedance measurements.</u> Once you have chosen a current probe of the correct current rating and properly terminated it, the next area of concern is the range setting of the current probe amplifier. The Tektronix current probe amplifier has a dynamic range of about ±10 divisions. In other words, if you set the current probe amplifier to 1 amp per division, it will read accurately up to about ±10 amps. In general, the measurements we are making are on DC power lines. We are superimposing a small amount of AC voltage in series with the DC output voltage of the source in order to make a small AC deviation in the DC current being drawn or supplied. The current probe amplifier range should be set so the DC current represents a large portion of the dynamic range but does not exceed the dynamic range. <u>For best accuracy, choose a range setting where the DC current being sensed represents between 3 and 8 divisions.</u> For example, if the DC current is 3 amperes, the current probe amplifier setting can be 1 amp per division (3 divisions) or 0.5 amps per division (6 divisions). Either setting will work and give good sensitivity without saturating the current probe amplifier.

The next critical adjustment of a current probe amplifier is DC balance. The newer current probe amplifiers are self-balancing, but older ones need to be adjusted so they output 0 volts DC when there is no current flowing through the probe. A small amount of DC error will not cause a problem since the FRA rejects DC, but if there is a large balance error it can affect the dynamic range of the amplifier and cause it to saturate with less than 10 divisions of deflection. Current probes get "gaussed", which means that they have some residual flux in the core of the transformer used to sense current. The DC balance adjustment corrects for this residual flux. Current probe amplifiers have a "Degauss" button, which drives the internal transformer with a large amplitude but exponentially decreasing AC excitation with the objective of minimizing the residual flux in the transformer. After you "degauss" the current probe, you will have to readjust the DC balance setting. The problem is that when you make another measurement, the DC current flowing through the probe "gausses" the transformer again and it immediately needs to be readjusted for DC balance. Our recommendation is to not degauss the probe because you will only make it difficult to adjust the DC balance again once you start taking measurements. One highly recommended step is to connect an oscilloscope in parallel with the FRA input (after the 50 ohm termination) so you can see the level of DC offset, the amount of deflection caused by the DC current, and the amount of AC deviation superimposed on the DC current.

The final detail of using a current probe is to calculate the effective resistance of the probe. If you used a resistor of value R to measure current, you would get R volts across the resistor for every amp through it. The current probe has an effective resistance, which is the number of volts out of the current probe amplifier for every amp through the current probe. The Tektronix AM503 series of current probe amplifiers have a fixed output of 0.01 volts for a unit of current equal to the range setting.

AM 503 Current Probe Amplifier

- •   Current Probe Output Voltage Scale = 10 mV/Div
- •   Current Probe Input Current Range = 1mA-50A/Div

The current probe amplifier output is 0.01 volts per division and the range setting is given in amps per division so the effective resistance of the current probe is 0.01 divided by the range setting. If the amplifier is set to 10 milliamps (0.01 amps) per division, then the effective resistance is 1 ohm. One common error in making impedance measurements is to leave the current probe amplifier set to 10 milliamps per division no matter what current is being measured. This simplifies the math, since there is no need to use the Scale Factor provision of the testing and plotting software. It also means that often the current probe will be operated far from its optimum range, sometimes even in the saturation region where the output is no longer linearly proportional to the input. <u>Failure to choose the proper range is another common source of error in impedance measurements.</u> That about covers the common mistakes people make in the use of current probes.

Calculating Scale Factor or Effective Resistance

- AM503 Scale Factor = Voltage Scale/Current Range

- Example
     10 mV/Div/50mA/Div = 0.2Ω
     Scale Factor or Effective Resistance = 0.2Ω

Now let's go to the subject of drive levels. There are two variables in setting the drive level. One is the level of the Oscillator Output of the FRA; the other is the gain setting of the power amplifier. Since the amplifier output is the product of these two gain settings, there is an infinite number of gain setting possibilities for any given output level. The simplest way to set the level is to use one of the gain variables to set the injection transformer output level to the desired level at an intermediate frequency such as 1 kHz and then leave the gain settings alone during the sweep. An effective way to do this adjustment is to set the FRA Oscillator Output to a reasonable level like 1 volt and then use a voltmeter or oscilloscope to monitor the output of the injection transformer and adjust the gain knob on the power amplifier for the desired output value. This can be done before power is applied; in fact it can be done before the injection transformer is even connected into the circuit. For example, let's say you are testing the impedance of some piece of the International Space Station. The Space Station power bus is 120 VDC and program officials have decided that a 2% variation in the power bus will not damage the equipment. That means that the 120 VDC bus can vary by ±2.4 volts and still be in the safe region. You can set the analyzer AC Volts Out to 1 volt peak. You can then monitor the output of the injection transformer and set the gain control of the power amplifier for ±2 volts peak voltage at 1 kHz. This will give a small margin of safety (from 2 volts to 2.4 volts). The amplitude of the oscillator output is very flat with frequency. The gain of the power amplifier is also flat with frequency except where the output falls off, below 10 Hz and above 100 kHz. This means that by setting the amplitude in mid-range (1 kHz), the output amplitude will not exceed this value at any time during the sweep.

When you plot the test results, the variable called Scale Factor on the Data Set Properties window is the resistance used to measure the current or the effective resistance of the current probe. The polarity will be correct if the current probe arrow is pointing toward the Unit Under Test. If the current probe arrow is pointing away from the Unit Under Test, the Scale Factor will have to be entered as a negative value.

The graph below is actual test data of the input impedance of one of the Orbital Replacement Units of the International Space Station.
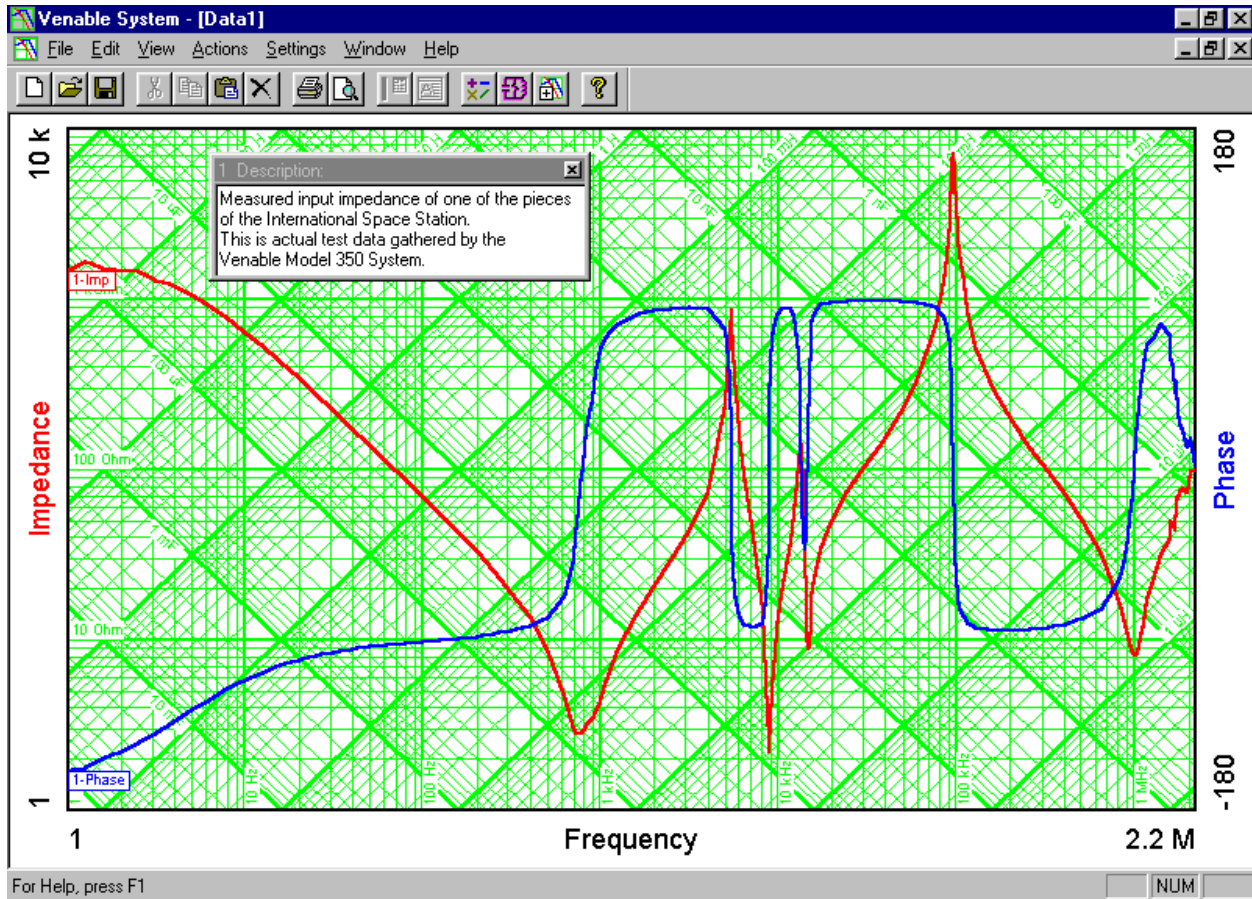
Circuit Modeling Overview

Introduction

The Venable software contains a built-in circuit-modeling program. This program is similar to SPICE in that it works from a net list. It is a relatively simple program. There are no libraries of parts and only AC analysis is performed. The advantage of this program is that it is very quick and easy to create and analyze models. It is easy to create accurate models of actual circuits since it is so easy to compare the model results with actual test results and also easy to change the model to make the model data match the test data.

We recognize that keying in a few lines of circuit description is not as glamorous as entering data with a schematic entry program, but we think you will find this method quicker and easier once you get used to it.

The following components are available:

V	Voltage source (volts)
I	Current source (amps)
R	Resistor (ohms)
L	Inductor (henries)
C	Capacitor (farads)
G	Conductance (mhos)

You can add annotation after any component designator. V, V213, or Vemf all will be interpreted as a voltage source.

The following abbreviations can be used for multipliers:

p or P	multiplied by 1E-12
n or N	multiplied by 1E-9
u or U	multiplied by 1E-6
m	multiplied by 1E-3
k or K	multiplied by 1E+3
M	multiplied by 1E+6
g or G	multiplied by 1E+9

The format of a source is:

Type of source	*positive control node*	*negative control node*	*value or gain*
Resistance	*positive connection node*	*negative connection node*	*value*

A special case is using 0 for both control nodes; this indicates a fixed source.

Spaces or commas separate entries.

Example 1: Voltage Source

V 0 0 1

R 1 0 50
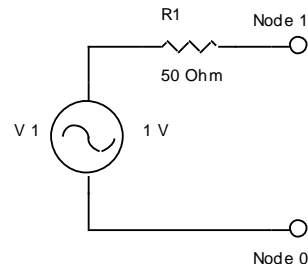
This is a fixed 1-volt source with an impedance of 50 ohms, connected in the circuit from node 1 (positive node) to node 0 (negative node). Node 0 is always circuit common.

Example 2: Current Source

I 3 4 0.1

R 2 0 1M
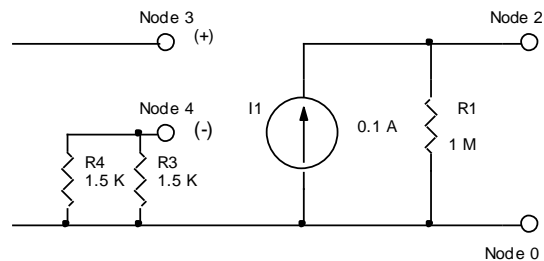
This is a current source controlled by the voltage from node 3 to node 4. It has a gain of 0.1, which means 1 volt from node 3 to node 4 will produce 0.1 amp from this source. It is connected in the circuit from node 2 to node 0, and is paralleled by a 1 mega-ohm resistor. Node 2 is in phase with node 3.

The format of a component is:

Component type	*connection node*	*connection node*	*value*

Example 3: Component (in this case, a capacitor)

C13 4 3 10U

This is a capacitor connected from node 4 to node 3 with a value of 10 microfarads. The order of the connection nodes is not significant. The number "13" after the C is any desired notation to identify the component, usually the reference designator.

Notes on model file creation:

Model files are ordinary ASCII text files. They can be created with any ASCII editing program such as Notepad in Windows.

Model files can have any extension, but if they have no extension a .CKT extension will be added when the circuit is saved.

Comment lines can be added at the beginning of a file by beginning each line with an asterisk.

Comments can be added at the end of any line. It is not necessary to begin a comment with an asterisk. The program will ignore anything on a line after reading the required four entries.

Blank lines are ignored.

Circuit node numbers must be sequential. They do not have to be entered in sequence, but all nodes from 0 to the highest number node must be present. A missing node number will generate an error.

Example 4: Circuit file for a low-pass filter

*Low-pass R-C filter

V1 0 0 1

R1 1 0 1

R2 1 2 1K

C1 2 0 0.1U
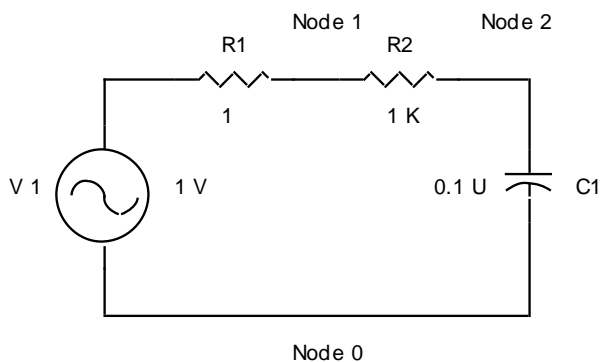
This circuit file represents a fixed one-volt source with a source impedance of 1 ohm driving a low-pass filter consisting of a 1,000 ohm resistor in series with a 0.1 microfarad capacitor to ground. When analyzed and plotted as the transfer function (Bode plot) from node 1 to node 2, the result will be a classic low-pass filter response. Since the result is plotted as the ratio of the voltage on node 2 divided by the voltage on node 1, the impedance of the source (1 ohm) does not enter into the calculation and the result will be the same no matter what the value of the source resistance.

Example 5: Measuring the output impedance of an input filter

*Model of input filter output impedance
V1 3 2 10   Controlled source translates current to ground reference. Gain=1/R3.
R1 1 0 1   Resistance of controlled source (value not significant)
V2 0 0 1   Fixed one volt source driving output of filter
R2 3 0 1   Impedance of test source (value not significant)
R3 3 2 0.1   Resistor to measure current. Different value changes gain of V1.
C1 2 4 100U   Output capacitance of L-C filter
R4 4 0 0.01   ESR of filter output capacitor
R5 2 5 2   Damping resistor
C2 5 0 390U   Capacitor to block DC current through damping resistor
L1 2 6 100U   Input inductance of L-C filter
R6 6 0 0.01   Resistance of input inductor
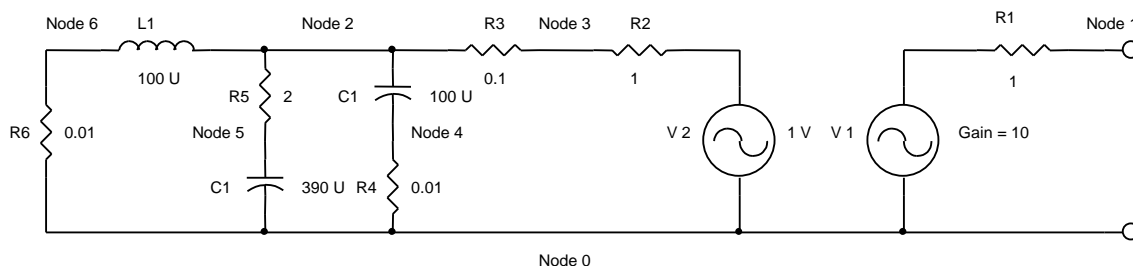
Normally R3 would be 1 ohm to simplify the calculation, and V1 would have a gain of 1. The values used in this example show the flexibility of the technique.

R6 is connected to ground to represent zero AC impedance of the power source.

In this example, filter impedance is measured as Node 2 over Node 1.

Note that the Venable system, unlike older Spice-type modeling programs, actually plots the ratio of the output to the input. Many older modeling programs require a one-volt source as an input and then plot absolute value of the output and pretend it is the ratio of output to input. This works when the input is exactly one volt, but the technique falls apart when the input is not one volt. When measuring gain by plotting the ratio of voltages on each side of an injection resistor, the capability to plot voltage ratios is essential.

APPENDIX A

"Option" Selection for model series 43xx/63xx/74xx/88xx/350c

The "Option" selection on the analyzer control menu for these analyzers, provide special capabilities and/or algorithms to be utilized during analysis under conditions that require them. Each option is toggled ON or OFF each time the option is selected. New capabilities and algorithms will be added to the option list as they are developed and become available by Venable Industries. These new options will be provided through firmware updates for the Venable Analyzer and updates to the Venable software. Details for each option are described below.

350b Operating Mode

This setting enables or disables continuous sine wave output through a measurement sweep. This provides analyzer operation identical to the legacy Venable 350 analyzers. This function is not compatible with the Digital Interface function. "Circuit under Test" must be set to DC to utilize this function.

Sync Output

This setting activates or deactivates the sync output on the back panel of the analyzer. The output produces a square wave signal that is frequency and phase locked to the generator output.

Digital Interface

This setting enables or disables the Digital I/O port on the 88xx series of analyzers. When the setting is turned OFF, the analyzer operates as a standard 2 channel analog analyzer. This function is not compatible with the 350b Operating Mode.

High Noise Algorithm

This setting engages algorithms within the analyzer itself, to provide improved results when connected to units under test that have high levels of noise inherent in them. This is typically when the noise voltage levels are more than 10X the injection voltage. ***Applications that do not experience these levels of noise should not use this mode.*** This algorithm generally increases bias elimination times and adjusts the criteria for auto-ranging to determine the most suitable range for analysis to provide better results. Integration time selection are not affected by this algorithm, however the user should expected much longer processing times to occur between frequency points.

"Circuit Under Test" Selection for model series 43xx/63xx/74xx/88xx/350c

This scroll box sets the operational state of the instrument to optimize its operation when channel inputs are attached to an AC environment. The user can select the line frequency (Hz) of the environment that the analyzer channels are being attached to. For measurement of DC power supplies, this selection should remain set to DC. When the 350b Operating Mode is active, only DC mode is allowed.

Notes: This capability is available in the following application firmware versions only:

        Version 1.x: version 1.7 or later
        Version 2.x: version 2.5 or later
        Version 3.x: version 3.3 or later

APPENDIX B

Calibration Specifications for 5140, 63xx, 74xx, 88xx & 350c Analyzers

Generator Output Tests: Test 10

DC Bias:　　Resolution:　10mV
　　　　　　 Accuracy:　　 +/- 1.0% of Range Full Scale (10VDC)

Square Wave Accuracy:　+/- 1.0% of Range Full Scale　(all ranges except 10mV)

　　　　　　　　　　　　　 +/- 1.5% of Range Full Scale　(10mV range only)

Output ranges
Range
10 mVpk
31.6 mVpk
100 mVpk
316 mVpk
1 Vpk
3.16 Vpk
10 Vpk

Sine Wave AC Amplitude Accuracy:

　　+/-(1.0% + 1%/MHz) of Range Full Scale　(all ranges except 10mV & 30mV)

　　+/-(1.8% + 1%/MHz) of Range Full Scale　(30mV range only)

　　+/-(2.6% + 1%/MHz) of Range Full Scale　(10mV range only)

　　　　Resolution:　Depends on the range setting (see table below)

Output ranges and resolution	
Range	*Sensitivity*
10 mVpk	10uV
31.6 mVpk	30uV
100 mVpk	100uV
316 mVpk	300uV
1 Vpk	1 mV
3.16 Vpk	3 mV
10 Vpk	10 mV

Channel Input Tests

Amplitude Accuracy: Tests 20 - 26

>= 1 MHz: +/- (1.0% + 1.3%/MHz) of Range Full Scale (all ranges)

 < 1 MHz: +/- (0.4% + 1.3%/MHz) of Range Full Scale (all ranges except 10mV)

 < 1 MHz: +/- (0.6% + 1.6%/MHz) of Range Full Scale (10mV range only)

Output ranges

Range
10 mVpk
31.6 mVpk
100 mVpk
316 mVpk
1 Vpk
3.16 Vpk
10 Vpk
31.6 Vpk
100 Vpk
316 Vpk
500 Vpk

Phase Accuracy: Tests 30 - 59

   +/- (0.4deg + 1.0 deg/MHz)

Bandwidth Accuracy: Tests 60 - 65

   Amplitude: +/- (0.04 dB + 0.1 dB /MHz)
   Phase: +/- (0.4 deg + 1.0 deg/MHz)

04/14/20

RLC Specifications

Measurement Frequency

1 Hz to 40 MHz, user-selectable for single component, selected automatically for multiple-component circuit.

Accuracy

1 Hz to 1 kHz	0.1%
1 kHz to 100 kHz	0.2%
100 kHz to 1 MHz	0.5%
1 MHz to 10 MHz	1.0%
10 MHz to 40 MHz	4.0%

Optimum Measurement Range

Resistance:	10 milliohms to 1 megaohm
Inductance:	1 microhenry to 10,000 henries
Capacitance:	100 picofarads to 1 farad

*Extended Measurement Range**

Resistance:	100 micro-ohms to 100 megaohms
Inductance:	10 nanohenrys to 1 megahenry
Capacitance:	1 picofarad to 100 farads

* Extended range loses least-significant digit of accuracy for each decade of measured value outside the optimum measurement or frequency range.

Specifications: Venable 350c, 3 channel, 5, 20, and 40MHz Models

Generator:

Frequency Range:	10µHz to 5, 20, or 40MHz (sine wave)
	10µHz to 1MHz (square wave)
AC Amplitude	1mV to 10V
DC Bias	±10V, 10mV Steps
Sweep Modes:	Single Frequency, logarithmic and linear sweep steps
	Log Sweep: 0.1 – 2000 Steps per decade
	Linear Sweep: 1Hz – 5, 20, or 40MHz step
Output Amplitude Compression:	Dynamically adjust output to maintain a constant channel input level through Venable software servo
Output Impedance:	Switchable: 50 ohms (default) or high current 2 ohms to 5 MHz
Output Configuration:	Single-ended floating and isolated up to ±600 Vpk

Analyzer:

Frequency Range:	10µHz to 5, 20, or 40MHz
Input Configuration:	Ch.1, 2, and 3: Single-ended Floating to ±600Vpk
Input Impedance:	Switchable: 50 Ohms or 1 Mega Ohm (default)
Measurement Accuracy:	± 0.03dB + .1dB/MHz;
	± 0.4deg + 1deg/MHz
Measurement Technique:	Narrowband DFT
	Integration Time: 20msec to 100ksec
	Delay Time: 0-100ksec
	Integration Cycles: 1-9999 cycles
	Delay Cycles: 1-9999 cycles
Input Coupling:	DC with automatic DC offset cancellation
Input Range:	10mV to 500Vpk Full Scale in 11 ranges, Auto-ranging
Dynamic Range:	120dB
CMRR/IMRR:	120dB
Max. Input:	±500Vpk
Max Input Withstand Voltage:	±600Vpk
Over-range Alarms:	LED indicator
Calibration:	NIST and Z540.1 standards

System:

PC Interface:	USB 2.0 and IEEE 488 (GPIB)
Auxiliary Output:	12Vdc @ 400mA, 4.8W for accessories
Application software:	Venable Stability Analysis™ for Win 7/8/10
Real time display update:	Each point is plotted as acquired
Data Analysis:	Polar coordinates: Gain magnitude, Phase, Gain margin, and phase margin
	Impedance: R, L, C, Z magnitude, and Phase
Power Supply Requirements:	90 to 264Vac, 48 to 62Hz, 30VA,
Weight/Dimensions:	12 Lbs. / 17"x 10"x 3.5"

04/14/20

Specifications: Venable 5140, 2 channel, 40MHz

Generator:

Frequency Range:	1Hz to 40MHz (sine wave)
AC Amplitude	10mV to 10V
DC Bias	±10V, 10mV Steps
Sweep Modes:	Single Frequency, logarithmic and linear sweep steps
	Log Sweep: 0.1 – 2000 Steps per decade
	Linear Sweep: 1Hz – 40MHz step
Output Amplitude Compression:	Dynamically adjust output to maintain a constant channel input level through Venable software servo
Output Impedance:	50 ohms
Output Configuration:	Single-ended grounded

Analyzer:

Frequency Range:	1Hz to 40MHz
Input Configuration:	Ch.1: Single-ended Floating to ±600Vpk
	Ch.2: Single-ended grounded
Input Impedance:	1 Mega ohm
Measurement Accuracy:	± 0.03dB + .1dB/MHz;
	± 0.4deg + 1deg/MHz
Measurement Technique:	Narrowband DFT
	Integration Time: 20msec to 100ksec
	Delay Time: 0-100 sec
	Integration Cycles: 1-9999 cycles
	Delay Cycles: 1-9999 cycles
Input Coupling:	DC with automatic DC offset cancellation
Input Range:	10mV to 10Vpk Full Scale in 7 ranges, Auto-ranging
Dynamic Range:	120dB
CMRR/IMRR:	120dB
Max. Input:	±100Vpk
Max Input Withstand Voltage:	±100Vpk
Over-range Alarms:	LED indicator
Calibration:	NIST Standard

System:

PC Interface:	USB 2.0
Auxiliary Output:	12Vdc @ 400mA, 4.8W for accessories
Application software:	Venable Stability Analysis™ for Win 7/8/10
Real time display update:	Each point is plotted as acquired
Data Analysis:	Polar coordinates: Gain magnitude, Phase, Gain margin, and phase margin
	Impedance: R, L, C, Z magnitude, and Phase
Power Supply Requirements:	90 to 264Vac, 48 to 62Hz, 30VA,
	24VDC, 24W (Min)
Weight/Dimensions:	4 Lbs. / 9.8"x 9.8"x 3.2"

04/14/20

Specifications: Venable 63XX, 2 channel, 5, 20, and 40MHz Models

Generator:

Frequency Range:	10µHz to 5, 20, or 40MHz (sine wave)
	10µHz to 1MHz (square wave)
AC Amplitude	1mV to 10V
DC Bias	±10V, 10mV Steps
Sweep Modes:	Single Frequency, logarithmic and linear sweep steps
	Log Sweep: 0.1 – 2000 Steps per decade
	Linear Sweep: 1Hz – 5, 20, or 40MHz step
Output Amplitude Compression:	Dynamically adjust output to maintain a constant channel input level through Venable software servo
Output Impedance:	Switchable: 50 ohms (default) or high current 2 ohms to 5 MHz
Output Configuration:	Single-ended floating and isolated up to ±600 Vpk

Analyzer:

Frequency Range:	10µHz to 5, 20, or 40MHz
Input Configuration:	Ch.1 and 2: Single-ended Floating to ±600Vpk
Input Impedance:	Switchable: 50 Ohms or 1 Mega Ohm (default)
Measurement Accuracy:	± 0.03dB + .1dB/MHz;
	± 0.4deg + 1deg/MHz
Measurement Technique:	Narrowband DFT
	Integration Time: 20msec to 100ksec
	Delay Time: 0-100ksec
	Integration Cycles: 1-9999 cycles
	Delay Cycles: 1-9999 cycles
Input Coupling:	DC with automatic DC offset cancellation
Input Range:	10mV to 500Vpk Full Scale in 11 ranges, Auto-ranging
Dynamic Range:	120dB
CMRR/IMRR:	120dB
Max. Input:	±500Vpk
Max Input Withstand Voltage:	±600Vpk
Over-range Alarms:	LED indicator
Calibration:	NIST and Z540.1 standards

System:

PC Interface:	USB 2.0 and IEEE 488 (GPIB)
Auxiliary Output:	12Vdc @ 400mA, 4.8W for accessories
Application software:	Venable Stability Analysis™ for Win 7/8/10
Real time display update:	Each point is plotted as acquired
Data Analysis:	Polar coordinates: Gain magnitude, Phase, Gain margin, and phase margin
	Impedance: R, L, C, Z magnitude, and Phase
Power Supply Requirements:	90 to 264Vac, 48 to 62Hz, 30VA,
Weight/Dimensions:	12 Lbs. / 17"x 10"x 3.5"

04/14/20

Specifications: Venable 74XX, 4 channel, 5, 20, and 40MHz Models

Generator:

Frequency Range:	10µHz to 5, 20, or 40MHz (sine wave)
	10µHz to 1MHz (square wave)
AC Amplitude	1mV to 10V
DC Bias	±10V, 10mV Steps
Sweep Modes:	Single Frequency, logarithmic and linear sweep steps
	Log Sweep: 0.1 – 2000 Steps per decade
	Linear Sweep: 1Hz – 5, 20, or 40MHz step
Output Amplitude Compression:	Dynamically adjust output to maintain a constant channel input level through Venable software servo
Output Impedance:	Switchable: 50 ohms (default) or high current 2 ohms to 5 MHz
Output Configuration:	Single-ended floating and isolated up to ±600 Vpk

Analyzer:

Frequency Range:	10µHz to 5, 20, or 40MHz
Input Configuration:	Ch.1, 2, 3 and 4: Single-ended Floating to ±600Vpk
Input Impedance:	Switchable: 50 Ohms or 1 Mega Ohm (default)
Measurement Accuracy:	± 0.03dB + .1dB/MHz;
	± 0.4deg + 1deg/MHz
Measurement Technique:	Narrowband DFT
	Integration Time: 20msec to 100ksec
	Delay Time: 0-100ksec
	Integration Cycles: 1-9999 cycles
	Delay Cycles: 1-9999 cycles
Input Coupling:	DC with automatic DC offset cancellation
Input Range:	10mV to 500Vpk Full Scale in 11 ranges, Auto-ranging
Dynamic Range:	120dB
CMRR/IMRR:	120dB
Max. Input:	±500Vpk
Max Input Withstand Voltage:	±600Vpk
Over-range Alarms:	LED indicator
Calibration:	NIST and Z540.1 standards

System:

PC Interface:	USB 2.0 and IEEE 488 (GPIB)
Auxiliary Output:	12Vdc @ 400mA, 4.8W for accessories
Application software:	Venable Stability Analysis™ for Win 7/8/10
Real time display update:	Each point is plotted as acquired
Data Analysis:	Polar coordinates: Gain magnitude, Phase, Gain margin, and phase margin
	Impedance: R, L, C, Z magnitude, and Phase
Power Supply Requirements:	90 to 264Vac, 48 to 62Hz, 30VA,
Weight/Dimensions:	12 Lbs. / 17"x 10"x 3.5"

Specifications: Venable 88XX, 23 channel, 5, 20, and 40MHz Models

Generator:

Frequency Range:	10μHz to 5, 20, or 40MHz (sine wave)
	10μHz to 5MHz (square wave)
AC Amplitude	1mV to 10V
DC Bias	±10V, 10mV Steps
Sweep Modes:	Single Frequency, logarithmic and linear sweep steps
	Log Sweep: 0.1 – 2000 Steps per decade
	Linear Sweep: 1Hz – 5, 20, or 40MHz step
Output Amplitude Compression:	Dynamically adjust output to maintain a constant channel input level through Venable software servo
Output Impedance:	Switchable: 50 ohms (default) or high current 2 ohms to 5 MHz
Output Configuration:	Single-ended floating and isolated up to ±600 Vpk

Analyzer:

Frequency Range:	10μHz to 5, 20, or 40MHz
Input Configuration:	Ch.1 and 2: Single-ended Floating to ±600Vpk
Input Impedance:	Switchable: 50 Ohms or 1 Mega Ohm (default)
Measurement Accuracy:	± 0.03dB + .1dB/MHz;
	± 0.4deg + 1deg/MHz
Measurement Technique:	Narrowband DFT
	Integration Time: 20msec to 100ksec
	Delay Time: 0-999 sec
	Integration Cycles: 1-9999 cycles
	Delay Cycles: 1-9999 cycles
Input Coupling:	DC with automatic DC offset cancellation
Input Range:	10mV to 500Vpk Full Scale in 11 ranges, Auto-ranging
Dynamic Range:	120dB
CMRR/IMRR:	120dB
Max. Input:	±500Vpk
Max Input Withstand Voltage:	±600Vpk
Over-range Alarms:	LED indicator
Calibration:	NIST and Z540.1 standards

Digital Analyzer:

Frequency Range:	10mHz to 1MHz
Integration Cycles:	1-10 cycles
Interface:	I^2C and Sync Out (3.3V)
Supported Processors:	Contact Venable

System:

PC Interface:	USB 2.0 and IEEE 488 (GPIB)
Auxiliary Output:	12Vdc @ 400mA, 4.8W for accessories
Application software:	Venable Stability Analysis™ for Win 7/8/10
Real time display update:	Each point is plotted as acquired

04/14/20
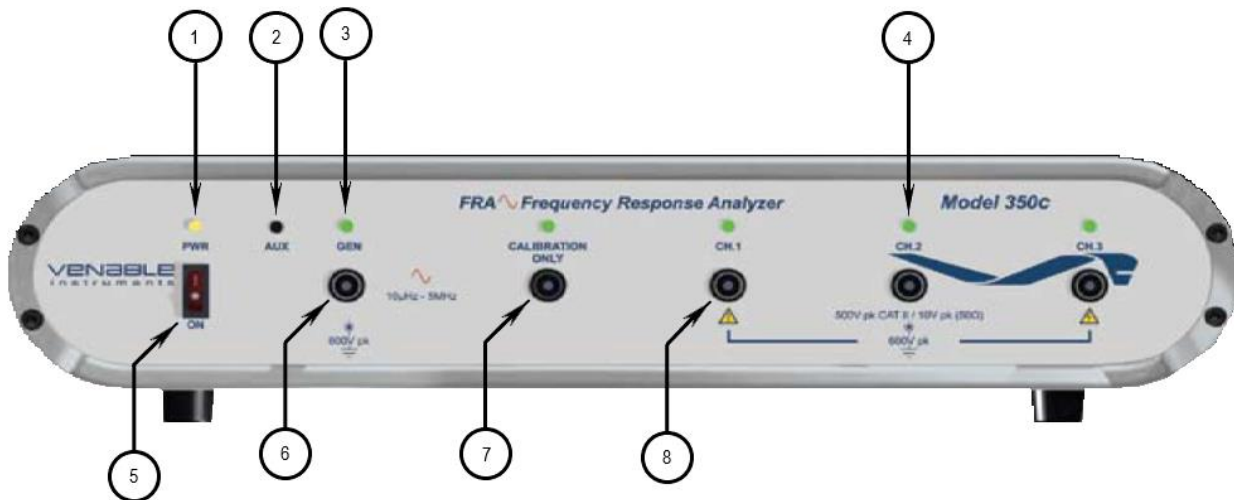
Data Analysis: Polar coordinates: Gain magnitude, Phase, Gain margin, and phase margin

Impedance: R, L, C, Z magnitude, and Phase

Power Supply Requirements: 90 to 264Vac, 48 to 62Hz, 30VA,

Weight/Dimensions: 12 Lbs. / 17"x 10"x 3.5"

APPENDIX C

## Hardware Description of 5140, 63xx, 74xx, 88xx & 350c Analyzers

350C Front Panel

Description

1. Power LED: This amber LED Illuminates when the power is turned on.
2. Auxiliary Power Jack: Provides 12Vdc@400mA, 4.8W for accessories such as the LF Bode Box.
3. Generator LED: This green LED will flash on and off when power is turned on.
                        It illuminates when the generator is turned on by the user.
                        Normally it is off.
4. Channel LED: This green LED will flash on and off when power is turned on. The
                        green LED illuminates when the analyzer has finished booting
                        up and indicates that the channel ready for measurements.
                        Normally it is on. A flickering LED indicates that the channel is
                        overloading and trying to clear the overload. If it is off, the
                        channel is malfunctioning or can't clear the overload.
5. Power Switch: This switch turns the analyzer on and off.
6. Generator Output: This is a floating single-ended output. .The output impedance is switchable
                        between 50 Ohms and 2 Ohms (high current). It is isolated and floats to
                        ±600Vpk.
7. Reference Channel: This channel is not used for measurement. It is used to measure the
                        oscillator output.
8. Input Channels: All three channels have single-ended floating inputs switchable between
                        input impedances of 50 Ohms and 1 Mega Ohm (default). They are
                        isolated and float to ±600Vpk.

350C/63XX/74XX/88XX Rear Panel
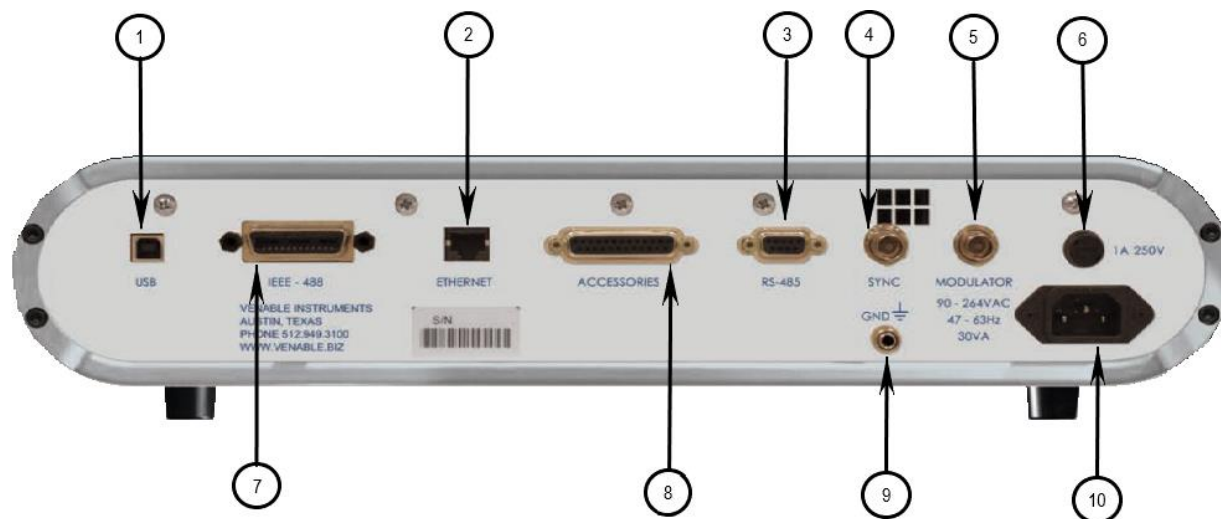
Description

1. USB Port: Provides USB 2.0 communication with the Venable software version 5 using the Keysight VISA or Venable software version 6 using the Windows USB driver (Win 7 and above).
2. Ethernet Port   This port is primarily used to upgrade the analyzer firmware.
3. RS-485 Port   Not implemented.
4. Sync Output   Outputs a CMOS level square wave signal set to the same frequency as the generator output.
5. Modulator Input   Not implemented.
6. Fuse   Fuse holder for 1A/250 slow-blow fuse.
7. GPIB Port: Provides IEEE 488 communication for Venable software and test systems.
8. Accessories Port: DB25 connector to control external accessories and the automated calibration test fixture.
9. Chassis Ground: Ground terminal that works with bare wire, lugs or banana plug
10. Power Inlet: Supplies the required 90 to 264Vac, 48 to 62Hz, and 30VA power to the analyzer.

04/14/20

5140 Front Panel

Description

1. Power LED: This amber LED Illuminates when the power is turned on.
2. Auxiliary Power Jack: Provides 12Vdc@400mA, 4.8W for accessories such as the LF Bode Box.
3. Generator LED: This green LED will flash on and off when power is turned on. It illuminates when the generator is turned on by the user. Normally it is off.
4. Channel LED: This green LED will flash on and off when power is turned on. The green LED illuminates when the analyzer has finished booting up and indicates that the channel ready for measurements. Normally it is on. A flickering LED indicates that the channel is overloading and trying to clear the overload. If it is off, the channel is malfunctioning or can't clear the overload.
5. Power Switch: This switch turns the analyzer on and off.
6. Generator Output: This is a grounded single-ended 50 Ohm output. It requires that an injection transformer be used when measuring feedback loops.
7. Channel 1 Input: This is a single-ended floating 1 Mega Ohm input. It is isolated and floats to ±600Vpk.
8. Channel 2 Input: This is a grounded single-ended 1 Mega Ohm input

5140 Rear Panel

Description

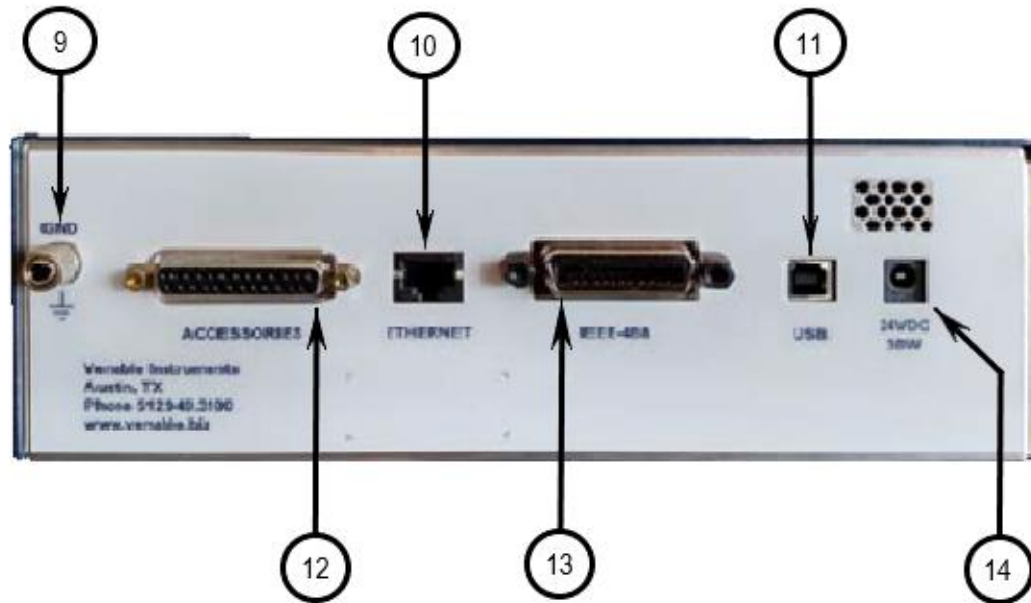|		
9.	Chassis Ground:	Ground terminal that works with bare wire, lugs or banana plug.
10.	Ethernet Port:	This port is primarily used to upgrade analyzer firmware.
11.	USB Port:	Provides USB 2.0 communication with the Venable software version 6 using the Windows USB driver (Win 7 and above).
12.	Accessories Port:	DB25 connector to control external accessories and the automated calibration test fixture.
13.	GPIB Port:	Provides IEEE 488 communication for test systems.
14.	DC Power Inlet:	Supplies the required 24VDC voltage and 24W (Min) power to the analyzer.

63XX Front Panel

Description

1. Power LED: This amber LED Illuminates when the power is turned on.
2. Auxiliary Power Jack: Provides 12Vdc@400mA, 4.8W for accessories such as the LF Bode Box.
3. Generator LED: This green LED will flash on and off when power is turned on.
                           It illuminates when the generator is turned on by the user.
                           Normally it is off.
4. Channel LED: This green LED will flash on and off when power is turned on. The
                           green LED illuminates when the analyzer has finished booting
                           up and indicates that the channel ready for measurements.
                           Normally it is on. A flickering LED indicates that the channel is
                           overloading and trying to clear the overload. If it is off, the
                           channel is malfunctioning or can't clear the overload.
5. Power Switch: This switch turns the analyzer on and off.
6. Generator Output: This is a floating single-ended output. .The output impedance is switchable
                           between 50 Ohms and 2 Ohms (high current). It is isolated and floats to
                           ±600Vpk.
7. Input Channels: Both channels have single-ended floating inputs switchable between input
                           impedances of 50 Ohms and 1 Mega Ohm (default). They are isolated and
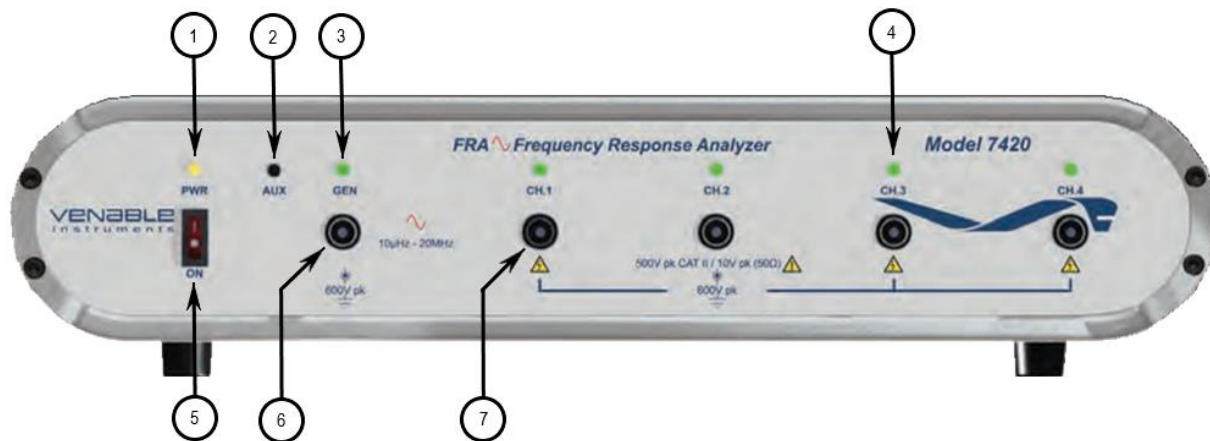   float to ±600Vpk.

74XX Front Panel

Description

1. Power LED: This amber LED Illuminates when the power is turned on.
2. Auxiliary Power Jack: Provides 12Vdc@400mA, 4.8W for accessories such as the LF Bode Box.
3. Generator LED: This green LED will flash on and off when power is turned on.
                         It illuminates when the generator is turned on by the user.
                         Normally it is off.
4. Channel LED: This green LED will flash on and off when power is turned on. The
                         green LED illuminates when the analyzer has finished booting
                         up and indicates that the channel ready for measurements.
                         Normally it is on. A flickering LED indicates that the channel is
                         overloading and trying to clear the overload. If it is off, the
                         channel is malfunctioning or can't clear the overload.
5. Power Switch: This switch turns the analyzer on and off.
6. Generator Output: This is a floating single-ended output. .The output impedance is switchable
                         between 50 Ohms and 2 Ohms (high current). It is isolated and floats to
                         ±600Vpk.
7. Input Channels: All four channels have single-ended floating inputs switchable between
                         input impedances of 50 Ohms and 1 Mega Ohm (default). They are
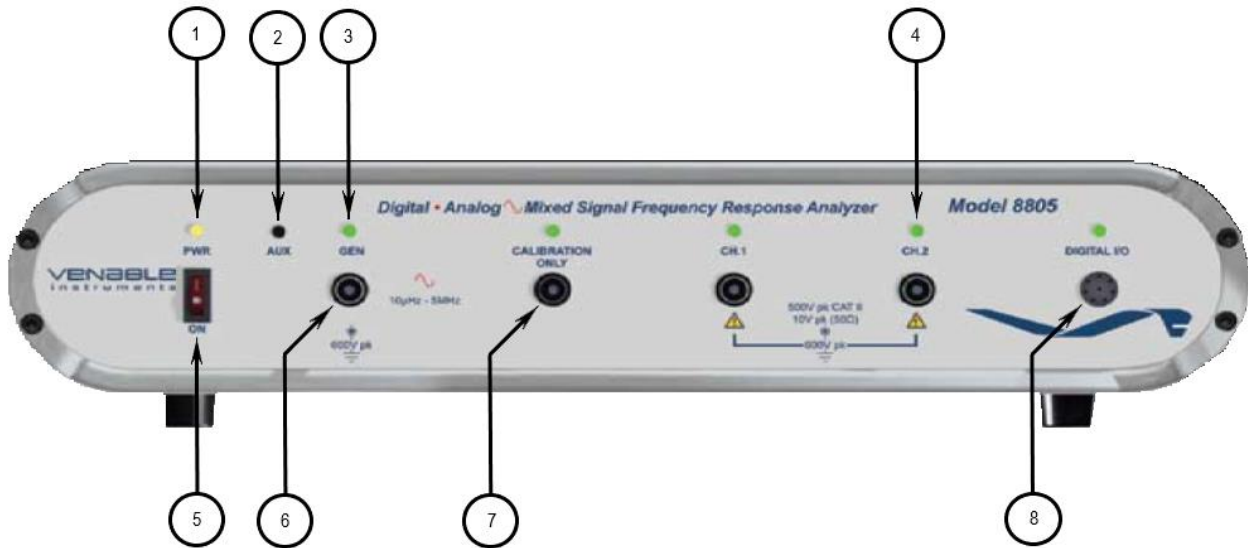   isolated and float to ±600Vpk.

88XX Front Panel

Description

1. Power LED: This amber LED Illuminates when the power is turned on.
2. Auxiliary Power Jack: Provides 12Vdc@400mA, 4.8W for accessories such as the LF Bode Box.
3. Generator LED: This green LED will flash on and off when power is turned on.
                          It illuminates when the generator is turned on by the user.
                          Normally it is off.
4. Channel LED: This green LED will flash on and off when power is turned on. The
                          green LED illuminates when the analyzer has finished booting
                          up and indicates that the channel ready for measurements.
                          Normally it is on. A flickering LED indicates that the channel is
                          overloading and trying to clear the overload. If it is off, the
                          channel is malfunctioning or can't clear the overload.
5. Power Switch: This switch turns the analyzer on and off.
6. Generator Output: This is a floating single-ended output. .The output impedance is switchable
                          between 50 Ohms and 2 Ohms (high current). It is isolated and floats to
                          ±600Vpk.
7. Input Channels: Both analog channels have single-ended floating inputs switchable
                          between input impedances of 50 Ohms and 1 Mega Ohm (default). They
                          are isolated and float to ±600Vpk.
8. Digital I/O Port: This port used to connect to a target processor in a digital power supply.
                          A special cable is used and the connection consists of an I2C bus, a sync
                          signal and a ground.